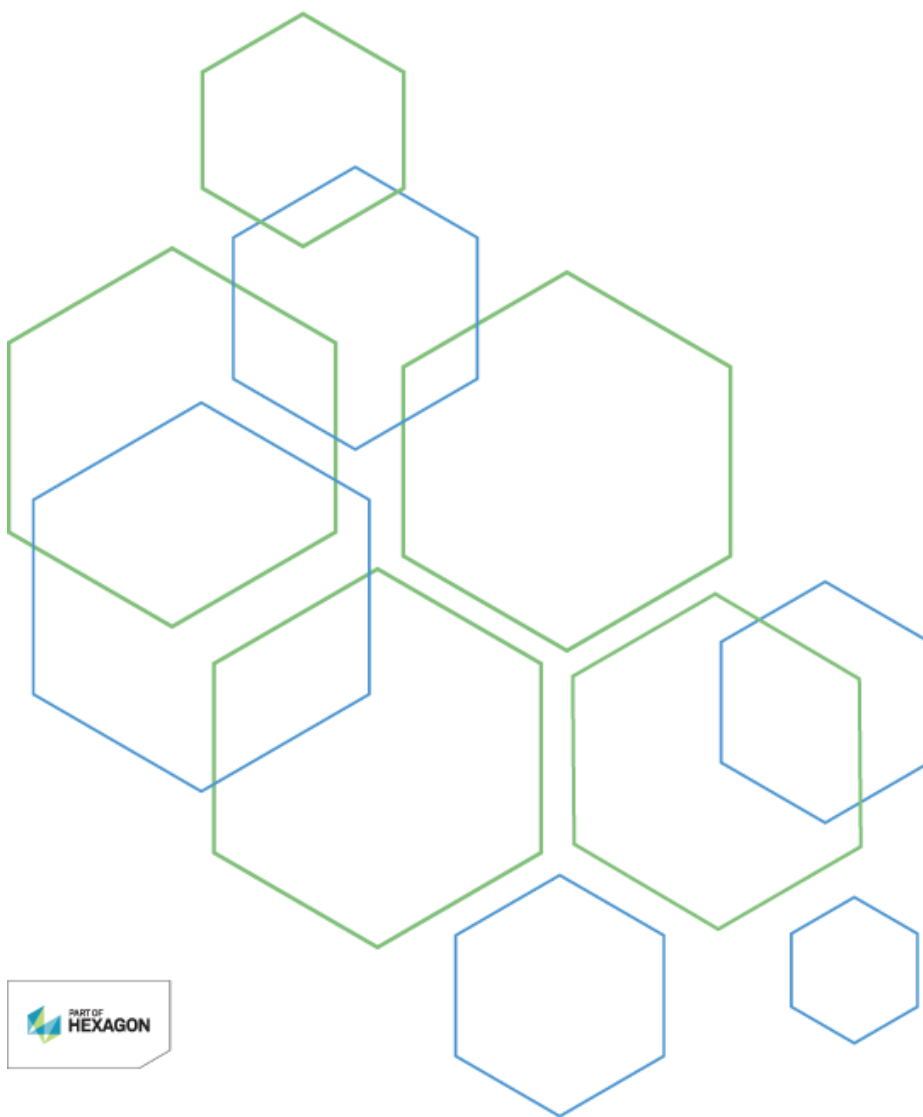


INTERGRAPH®  
**Smart** ➞ **3D**  
Structural Analysis  
User's Guide



Version 2016 (11.0)  
November 2016

## Copyright

Copyright © 2004-2016 Intergraph® Corporation. All Rights Reserved. Intergraph is part of **Hexagon**.

Including software, file formats, and audiovisual displays; may be used pursuant to applicable software license agreement; contains confidential and proprietary information of Intergraph and/or third parties which is protected by copyright law, trade secret law, and international treaty, and may not be provided or otherwise made available without proper authorization from Intergraph Corporation.

Portions of this software are owned by Spatial Corp. © 1986-2016. All Rights Reserved.

Portions of the user interface are copyright © 2012-2016 Telerik AD.

## U.S. Government Restricted Rights Legend

Use, duplication, or disclosure by the government is subject to restrictions as set forth below. For civilian agencies: This was developed at private expense and is "restricted computer software" submitted with restricted rights in accordance with subparagraphs (a) through (d) of the Commercial Computer Software - Restricted Rights clause at 52.227-19 of the Federal Acquisition Regulations ("FAR") and its successors, and is unpublished and all rights are reserved under the copyright laws of the United States. For units of the Department of Defense ("DoD"): This is "commercial computer software" as defined at DFARS 252.227-7014 and the rights of the Government are as specified at DFARS 227.7202-3.

Unpublished - rights reserved under the copyright laws of the United States.

Intergraph Corporation  
305 Intergraph Way  
Madison, AL 35758

## Documentation

Documentation shall mean, whether in electronic or printed form, User's Guides, Installation Guides, Reference Guides, Administrator's Guides, Customization Guides, Programmer's Guides, Configuration Guides and Help Guides delivered with a particular software product.

## Other Documentation

Other Documentation shall mean, whether in electronic or printed form and delivered with software or on Intergraph Smart Support, SharePoint, or box.net, any documentation related to work processes, workflows, and best practices that is provided by Intergraph as guidance for using a software product.

## Terms of Use

- a. Use of a software product and Documentation is subject to the End User License Agreement ("EULA") delivered with the software product unless the Licensee has a valid signed license for this software product with Intergraph Corporation. If the Licensee has a valid signed license for this software product with Intergraph Corporation, the valid signed license shall take precedence and govern the use of this software product and Documentation. Subject to the terms contained within the applicable license agreement, Intergraph Corporation gives Licensee permission to print a reasonable number of copies of the Documentation as defined in the applicable license agreement and delivered with the software product for Licensee's internal, non-commercial use. The Documentation may not be printed for resale or redistribution.
- b. For use of Documentation or Other Documentation where end user does not receive a EULA or does not have a valid license agreement with Intergraph, Intergraph grants the Licensee a non-exclusive license to use the Documentation or Other Documentation for Licensee's internal non-commercial use. Intergraph Corporation gives Licensee permission to print a reasonable number of copies of Other Documentation for Licensee's internal, non-commercial use. The Other Documentation may not be printed for resale or redistribution. This license contained in this subsection b) may be terminated at any time and for any reason by Intergraph Corporation by giving written notice to Licensee.

## Disclaimer of Warranties

Except for any express warranties as may be stated in the EULA or separate license or separate terms and conditions, Intergraph Corporation disclaims any and all express or implied warranties including, but not limited to the implied warranties of merchantability and fitness for a particular purpose and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such disclaimer. Intergraph believes the information in this publication is accurate as of its publication date.

The information and the software discussed in this document are subject to change without notice and are subject to applicable technical product descriptions. Intergraph Corporation is not responsible for any error that may appear in this document.

The software, Documentation and Other Documentation discussed in this document are furnished under a license and may be used or copied only in accordance with the terms of this license. THE USER OF THE SOFTWARE IS EXPECTED TO MAKE THE FINAL EVALUATION AS TO THE USEFULNESS OF THE SOFTWARE IN HIS OWN ENVIRONMENT.

Intergraph is not responsible for the accuracy of delivered data including, but not limited to, catalog, reference and symbol data. Users should verify for themselves that the data is accurate and suitable for their project work.

## Limitation of Damages

IN NO EVENT WILL INTERGRAPH CORPORATION BE LIABLE FOR ANY DIRECT, INDIRECT, CONSEQUENTIAL INCIDENTAL, SPECIAL, OR PUNITIVE DAMAGES, INCLUDING BUT NOT LIMITED TO, LOSS OF USE OR PRODUCTION, LOSS OF REVENUE OR PROFIT, LOSS OF DATA, OR CLAIMS OF THIRD PARTIES, EVEN IF INTERGRAPH CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

UNDER NO CIRCUMSTANCES SHALL INTERGRAPH CORPORATION'S LIABILITY EXCEED THE AMOUNT THAT INTERGRAPH CORPORATION HAS BEEN PAID BY LICENSEE UNDER THIS AGREEMENT AT THE TIME THE CLAIM IS MADE. EXCEPT WHERE PROHIBITED BY APPLICABLE LAW, NO CLAIM, REGARDLESS OF FORM, ARISING OUT OF OR IN CONNECTION WITH THE SUBJECT MATTER OF THIS DOCUMENT MAY BE BROUGHT BY LICENSEE MORE THAN TWO (2) YEARS AFTER THE EVENT GIVING RISE TO THE CAUSE OF ACTION HAS OCCURRED.

IF UNDER THE LAW RULED APPLICABLE ANY PART OF THIS SECTION IS INVALID, THEN INTERGRAPH LIMITS ITS LIABILITY TO THE MAXIMUM EXTENT ALLOWED BY SAID LAW.

## Export Controls

Intergraph Corporation's software products and any third-party Software Products obtained from Intergraph Corporation, its subsidiaries, or distributors (including any Documentation, Other Documentation or technical data related to these products) are subject to the export control laws and regulations of the United States. Diversion contrary to U.S. law is prohibited. These Software Products, and the direct product thereof, must not be exported or re-exported, directly or indirectly (including via remote access) under the following circumstances:

- a. To Cuba, Iran, North Korea, Sudan, or Syria, or any national of these countries.
- b. To any person or entity listed on any U.S. government denial list, including but not limited to, the U.S. Department of Commerce Denied Persons, Entities, and Unverified Lists, <http://www.bis.doc.gov/complianceand enforcement/liststocheck.htm>, the U.S. Department of Treasury Specially Designated Nationals List, <http://www.treas.gov/offices/enforcement/ofac/>, and the U.S. Department of State Debarred List, <http://www.pmddtc.state.gov/compliance/debar.html>.
- c. To any entity when Licensee knows, or has reason to know, the end use of the Software Product is related to the design, development, production, or use of missiles, chemical, biological, or nuclear weapons, or other un-safeguarded or sensitive nuclear uses.
- d. To any entity when Licensee knows, or has reason to know, that an illegal reshipment will take place.

Any questions regarding export or re-export of these Software Products should be addressed to Intergraph Corporation's Export Compliance Department, Huntsville, Alabama 35894, USA.

## Trademarks

Intergraph, the Intergraph logo, PDS, SmartPlant, FrameWorks, I-Sketch, SmartMarine, IntelliShip, ISOGEN, SmartSketch, SPOOLGEN, SupportManager, SupportModeler, Sapphire, and Intergraph Smart are trademarks or registered trademarks of Intergraph Corporation or its subsidiaries in the United States and other countries. Hexagon and the Hexagon logo are registered trademarks of Hexagon AB or its subsidiaries. Microsoft and Windows are registered trademarks of Microsoft Corporation. ACIS is a registered trademark of SPATIAL TECHNOLOGY, INC. Infragistics, Presentation Layer Framework, ActiveTreeView Ctrl, ProtoViewCtrl, ActiveThreed Ctrl, ActiveListBar Ctrl, ActiveSplitter, ActiveToolbars Ctrl, ActiveToolbars Plus Ctrl, and ProtoView are trademarks of Infragistics, Inc. Incorporates portions of 2D DCM, 3D DCM, and HLM by Siemens Product Lifecycle Management Software III (GB) Ltd. All rights reserved. Gigasoft is a registered trademark, and ProEssentials a trademark of Gigasoft, Inc. VideoSoft and VXFlexGrid are either registered trademarks or trademarks of ComponentOne LLC 1991-2013, All rights reserved. Oracle, JD Edwards, PeopleSoft, and Retek are registered trademarks of Oracle Corporation and/or its affiliates. Tribon is a trademark of AVEVA Group plc. Alma and act/cut are trademarks of the Alma company. Other brands and product names are trademarks of their respective owners.

# Contents

<b>Preface</b> .....	<b>6</b>
<b>What's New in Structural Analysis</b> .....	<b>6</b>
<b>Structural Analysis</b> .....	<b>7</b>
<b>Structural Analysis Workflow</b> .....	<b>8</b>
Structural Analysis Common Tasks .....	8
Selecting Objects .....	9
<b>New Analysis Models</b> .....	<b>11</b>
Create an analysis model .....	13
Update an analysis model.....	13
Change the analysis model member filter .....	13
Delete an analysis model.....	14
Create a new filter .....	14
Model Properties Dialog Box .....	16
General Tab (Model Properties Dialog Box) .....	16
Relationship Tab.....	17
Configuration Tab .....	18
Notes Tab .....	19
<b>New Mapping File</b> .....	<b>21</b>
Create a Mapping File.....	24
<b>Export Analytical Models</b> .....	<b>25</b>
Export analysis model .....	28
<b>Update from Analysis/Design</b> .....	<b>30</b>
Preview analysis/design results .....	33
Update analysis model using analysis/design results.....	33
<b>Loads</b> .....	<b>34</b>
<b>New Load Cases</b> .....	<b>37</b>
Create a load case .....	38
Delete a load case .....	38
Add a load case to a load combination .....	39
Load Case Properties Dialog Box.....	39
General Tab (Load Case Properties Dialog Box) .....	39

<b>New Load Combinations.....</b>	<b>40</b>
Create a load combination .....	41
Add a load case to a load combination .....	42
Remove a load case from a load combination.....	42
Delete a load combination .....	43
Load Combination Properties Dialog Box .....	43
General Tab (Load Combination Properties Dialog Box).....	43
Load Cases Tab (Load Combination Properties Dialog Box) .....	44
<b>Place New Concentrated Loads .....</b>	<b>45</b>
Place concentrated load by absolute key-in .....	47
Place concentrated load by relative key-in .....	48
Place concentrated load using point along .....	50
Modify concentrated load magnitude .....	52
Move a concentrated load.....	52
Delete a concentrated load .....	53
Concentrated Load Properties Dialog Box .....	53
General Tab (Concentrated Load Properties Dialog Box) .....	53
<b>Place New Distributed Loads .....</b>	<b>55</b>
Place a fully distributed load .....	57
Place a partially distributed load .....	58
Modify distributed load magnitude .....	59
Modify distributed load position.....	60
Delete a distributed load .....	61
Distributed Load Properties Dialog Box.....	61
General Tab (Distributed Load Properties Dialog Box).....	61
<b>Set Boundary Conditions .....</b>	<b>63</b>
Place fixed boundary condition .....	65
Place pinned boundary condition .....	65
Modify a boundary condition .....	65
Removing a boundary condition .....	66
Define end releases .....	66
Boundary Condition Properties Dialog Box .....	66
General Tab (Boundary Condition Properties Dialog Box) .....	67
<b>Glossary .....</b>	<b>68</b>
<b>Index .....</b>	<b>76</b>

# Preface

This document is a user's guide for the Structural Analysis functionality of Intergraph Smart™ 3D and provides command reference information and procedural instructions.

## Documentation Comments

For the latest support information for this product, comments or suggestions about this documentation, and documentation updates for supported software versions, please visit *Intergraph Smart Support* (<https://smartsupport.intergraph.com>).

## What's New in Structural Analysis

The following changes have been made to the Structural Analysis task.


*Version 2016 (11.0)*


- Added a new locate filter, **Construction Graphics**. For more information, see *Selecting Objects* (on page 9). (P2 CP:271166)


## SECTION 1


# Structural Analysis


The Structural Analysis task places and modifies pre-analysis objects such as load cases and load combinations, loads, and boundary conditions. Also in this task, you can use the model geometry created in Smart 3D to generate a CIMsteel Integration Standards Release 2 analytical model for structural analysis and design. Results from analysis and design can be imported back into the model for immediate update of the members' section sizes. The Structural Analysis task has these commands:


 **Select** - Used to select objects in the model. For more information, see *Selecting Objects* (on page 9).


 **New Analysis Model Command** - Creates an analysis model, which is a logical collection of member parts, boundary conditions, and load combinations to export. For more information, see *New Analysis Models* (on page 11).


 **Update from Analysis/Design** - Reads a CIMsteel Integration Standard (CIS/2 format) file that contains all the analysis and design results from a third party analysis and design program. For more information, see *Update from Analysis/Design* (on page 30).

 **New Load Case** - Creates and edits load cases. For more information, see *New Load Cases* (on page 37).

 **New Load Combination** - Creates and edits load combinations. For more information, see *New Load Combinations* (on page 40).

 **Place New Concentrated Load** - Places a load that acts on a small area of a member. For more information, see *Place New Concentrated Loads* (on page 45).

 **Place New Distributed Load** - Places a load distributed along the full or partial length of a member. For more information, see *Place New Distributed Loads* (on page 55).

 **Set Boundary Condition** - Defines the support conditions that occur between the end of a member and the ground. For more information, see *Set Boundary Conditions* (on page 63).

**Export Analytical Model** - Writes an analysis model to a CIMsteel Integration Standard (CIS/2 format) file to allow a third party program to import the finite element representation of the analysis model. This command is on the **File** menu. For more information, see *Export Analytical Models* (on page 25).

**New Mapping File** - Creates an XML mapping file for the section and material names used in the software and third-party application. This command is on the **File** menu. For more information, see *New Mapping File* (on page 21).

## SECTION 2

# Structural Analysis Workflow

Because the loads and boundary conditions that you can place in the Structural Analysis task require structural members, you must first place the structural members using the Structure task. Select **Tasks > Structure** to switch to the Structure task.

When the structure is in place, use the commands found in this task to:

- Create an analysis model
- Define load cases, load combinations, and loads
- Define boundary conditions and member releases
- Create a CIS file to be analyzed and designed by a third-party application
- Update member section sizes based on the third-party application analysis and design

See Also

*Loads* (on page 34)

*Structural Analysis Common Tasks* (on page 8)

## Structural Analysis Common Tasks

The following tasks are used frequently in the structural analysis task.

### Create Analysis Model

All analytical data is gathered and exported based on analysis models. The first step in this task is to create an analysis model. For more information, see *Create an Analysis Model* (on page 13).

### Create Loads and Load Combinations

Create all the load cases that you plan to use. For more information, see *Create a Load Case* (on page 38).

Create all the load combinations that you plan to use. For more information, see *Create a Load Combination* (on page 41).

### Place Loads

Place loads on the members in the model. There are several methods for placing loads:

*Place a Fully Distributed Load* (on page 57)

*Place a Partially Distributed Load* (on page 58)

*Place Concentrated Load by Absolute Key-in* (on page 47)

*Place Concentrated Load by Relative Key-in* (on page 48)

*Place Concentrated Load using Point Along* (on page 50)



## Define Releases and Boundary Conditions

Define needed boundary conditions. For more information, see *Place Fixed Boundary Condition* (on page 65).

You can define member releases in the Structure task or in this task. For more information, see *Define End Releases* (on page 66).

## Create a Mapping File

Because third-party applications sometimes use different names for member section sizes and materials, you need to create a mapping file to prevent mismatches. For more information, see *Create a Mapping File* (on page 24).

## Export the Analytical Model

Export the Analysis Model for analysis and design. For more information, see *Export Analysis Model* (on page 28).

## Update Model with Analysis and Design Results

Update the section size of the members in the model with the analysis and design results. For more information, see *Update Analysis Model using Analysis/Design Results* (on page 33).

# Selecting Objects

All objects in the Structural Analysis task have properties that you can edit. Using the **Select** command on the vertical toolbar, you select the object to edit.



An important part of the **Select** command is the **Locate Filter** box that appears on the ribbon. The **Locate Filter** box contains the available, pre-defined filters for the **Select** command. When you choose a filter in the **Locate Filter** box, the software allows you to select only the filtered objects in a graphic view and in the **Workspace Explorer**. For example, if you select **Boundary Conditions**, you can select only boundary conditions, supports, in a graphic view or in the **Workspace Explorer**.

The Structural Analysis task includes these filters:

### Analysis Models

Limits your selection in the **Workspace Explorer** to analysis models.

### Boundary Conditions

Limits your selection in a graphic view or in the **Workspace Explorer** to boundary conditions.

### Construction Graphics

Limits the selection of items to construction graphics.

### Control Points

Limits the selection of items to the control points of an equipment object.

### Load Cases

Limits your selection in a graphic view or in the **Workspace Explorer** to load cases.

### Load Combinations

Limits your selection in a graphic view or in the **Workspace Explorer** to load combinations.

### Loads

Limits your selection in a graphic view or in the **Workspace Explorer** to loads.

### Member Parts

Limits your selection in a graphic view or in the **Workspace Explorer** to member parts.

### Structural Analysis

Allows you to select any object in a graphic view or in the Workspace Explorer that was placed using the Structural Analysis task. Objects placed using other tasks, such as equipment, cannot be selected using this filter.

### All

Allows you to select any object, even objects created in another task.



Use the **Inside** fence command to select all objects entirely inside the fence.




Use the **Inside/Overlapping** fence command to select all objects entirely inside the fence and those objects outside but touching the fence at some point.

### See Also

*Model Properties Dialog Box* (on page 16)

## SECTION 3

# New Analysis Models

 Creates a new analysis model. The exporting and importing of analytical data from your model is based on the analysis model, which is a non-graphical, logical grouping of member systems that you are going to send to an analysis and design program using a CIMsteel Integration Standard (CIS) file.

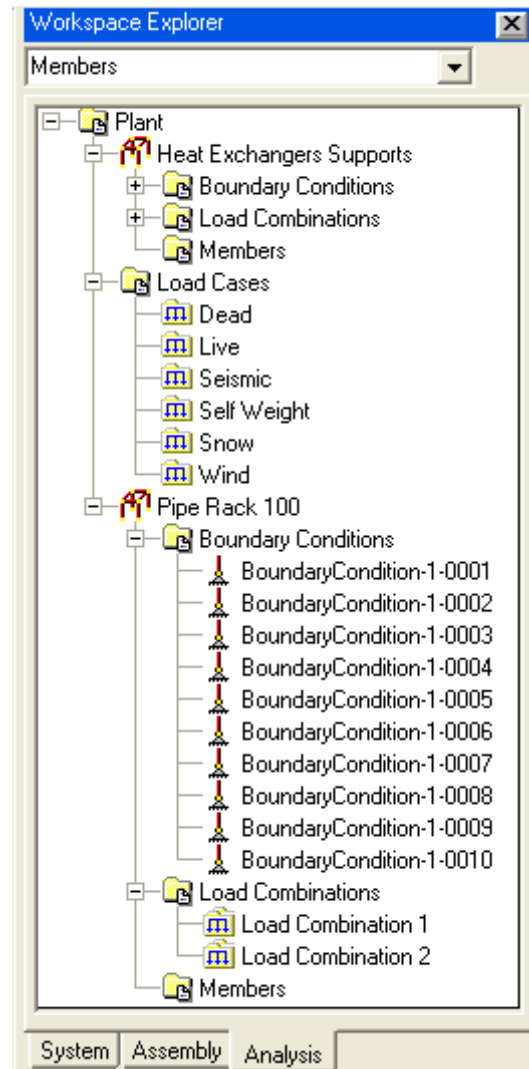
Analysis models appear on the **Analysis** tab of the **Workspace Explorer**. By default, the **Analysis** tab is not displayed in the **Workspace Explorer**. Click **Tools > Options**, and then select the **General** tab. In the **Tabs to display in Workspace Explorer** list, select the box next to **Analysis**. Save, exit, and reopen the session file to see the **Analysis** tab in the **Workspace Explorer**.

An analysis model has a fixed hierarchy of boundary conditions, load combinations, and members that belong to the analysis model. As you create boundary (support) conditions, loads, and load combinations, they appear underneath the respective categories on the **Analysis** tab.

Only member parts assigned to an analysis model are written to the CIS file. You assign member parts to the analysis model using a standard Model Filter. The filter looks for member systems, but it is the member parts of those filtered member systems that get assigned to the analysis model. This methodology ensures that intact member systems are written to the CIS file.

The filter is applied against the database to ensure that all member systems that pass the filter are found regardless of your current workspace definition. Remember that when a filter is associated to an analysis model, it is a picture of member systems that pass the filter at that time. As member systems are added or modified, the set of member systems that would pass the filter can change. You must re-apply the filter by modifying the analysis model to include and remove these members as needed. After you re-apply the filter, member systems with loads assigned to them that no longer pass the filter are removed from the analysis model.

You select (and therefore highlight in a view) all the members assigned to an analysis model by right-clicking on the Members folder and then selecting the **Select Nested** command.



Be sure to include objects on the **Analysis** tab when you define your *workspace filter*. Otherwise, your analysis model objects, except members, will disappear from the **Workspace Explorer** the next time you refresh your workspace. However, when defining your *selection filter* for the analysis model, make sure that you do *not* include any object on the **Analysis** tab of the **New Filter Properties** dialog box. Including objects on the **Analysis** tab in your selection filter causes all members with loads to be assigned to the analysis model whether they should be or not.

★ **IMPORTANT** You must create an analysis model before you can define load combinations or boundary conditions.

### Modify Analysis Model Ribbon

Displays options for modifying analysis models. You must select an analysis model on the **Analysis** tab of the **Workspace Explorer** for this ribbon to appear.

#### **Properties**

Opens the **Analysis Model Properties** dialog box that you can use to set additional analysis model properties that are not available on the ribbon. For more information, see *Model Properties Dialog Box* (on page 16).

#### **Selection Filter**

Select the Model filter used to identify the member systems to associate with the analysis model. When defining your selection filter, make sure that you do not include any object on the **Analysis** tab of the **New Filter Properties** dialog box. Including objects on the **Analysis** tab in your filter causes all members with loads to be assigned to the analysis model whether they should be or not.

#### **Apply Filter**

Updates the member systems (and therefore the member parts) assigned to the analysis model using the selected filter.

#### **Name**


Type a name for the analysis model.

---

### What do you want to do?

- *Create an analysis model* (on page 13)
  - *Update an analysis model* (on page 13)
  - *Change the analysis model member filter* (on page 13)
  - *Delete an analysis model* (on page 14)
-

## Create an analysis model


1. Click **New Analysis Model**  on the vertical toolbar.
2. In the **Name** box, type a name for the analysis model. If you do not specify a name, the software generates a name based on a naming rule.
3. Type a description for the analysis model.
4. Select a frame type and analysis method.
5. Select a coordinate system.
6. In the **Selection filter** box, select **Create New Filter** to create a filter for the members to include in the analysis model. When defining your filter, make sure that you do not include *any* object on the **Analysis** tab of the **New Filter Properties** dialog box. Including objects on the **Analysis** tab in your filter causes all members with loads to be assigned to the analysis model whether they should be or not.


*Create a New Filter* (on page 14)

 **TIP** Click **More** in the **Selection Filter** box to use an existing filter.


7. Click **OK**.

## Update an analysis model

1. Click **Select**  on the vertical toolbar.
2. Select **Analysis Models** in the **Locate Filter**.
3. Select the **Analysis** tab in the **Workspace Explorer**.
4. Select the analysis model to update.
5. Click **Apply Filter** on the ribbon.

 **NOTE** This procedure is for adding or removing member parts from the analysis models using the filter. To update members with analysis and design results, see *Update Analysis Model using Analysis/Design Results* (on page 33).

## Change the analysis model member filter


1. Click **Select**  on the vertical toolbar.
2. Select **Analysis Models** in the **Locate Filter**.
3. Select the **Analysis** tab in the **Workspace Explorer**.
4. Select the analysis model in **Workspace Explorer**.
5. In the **Selection filter** box on the ribbon, select a new filter.

*Create a New Filter* (on page 14)

 **TIP** Click **More** in the **Selection filter** box to use an existing filter.


6. Click **Apply Filter**.

## Delete an analysis model

1. Click **Select**  on the vertical toolbar.
2. Select **Analysis Models** in the **Locate Filter**.
3. Select the **Analysis** tab in the **Workspace Explorer**.
4. Right-click on the analysis model to delete, and then select **Delete**.

**NOTE** All load combinations and boundary conditions associated to the analysis model are removed and the associations to the member parts are dissolved. The member parts are not deleted from the model.

## Create a new filter

1. Open the **Select Filter** dialog box.
  - TIP** You can either click **Tools > Select by Filter**, or select **More** from the **Filter** list in the **Define Workspace** dialog box or the **Surface Style Rule** properties.
2. Click **New Filter**  to create a new filter. Make sure you select the type of filter from the following filters.
  - a. Select **Catalog Filters** or sub folder under that node to create a new catalog filter.
  - b. Select **Plant Filters** or sub folder under that node to create a new plant filter.
  - c. Select **My Filters** or sub folder under that node to create a new personal filter.
  - TIP** To create a new filter under **My Filters** folder, click **File > Define Workspace**, or press CTRL+W. In the **Plant** list, click the model in which you want to work. In the **Filters** list, click **Create New Filter**.
3. Type a name for the new filter in the **Name** box.
  - TIP** Filter names must be unique within the folder. You can have a filter in your **My Filters** folder with the same name as a filter in another folder. If you move a filter into a folder that already contains a filter of the same name, the software adds a numeric suffix to the filter name to keep the filter names unique.
4. Specify the filter search queries using one or more of the tabs on the **New Filter Properties** dialog box.
5. Select the **Include nested objects** option if you want the search to include all objects within a category. Otherwise, you must separately select each category and individual object.
6. Use the **System** tab to navigate the tree list to the systems to include in the query. These systems include the model root at the highest point of the hierarchy, as well as all subsystems, and specific types of objects. Select the **Include nested objects** option, if you want to include all objects under specified System nodes.
7. Use the **Assembly** tab to navigate the tree view and select the assemblies to include in the search. Select the **Include nested objects** option, if you want to include all objects under specified Assembly nodes.

8. Use the **Named Space** tab to indicate the named spaces and drawing volumes to include in the search. Select the **Include nested objects** option, if you want to include all objects under specified Space nodes.

#### TIPS

- A named space is a volume in the model, like a fire or blast zone.
  - A drawing volume defines the clipping volume associated with a specific drawing view in a document.
9. Use the **Analysis** tab to select structural analysis models to include in the search. Select the **Include nested objects** option, if you want to include all objects under specified Analysis entities.
  10. Use the **Work Breakdown Structure** tab to identify components in the Work Breakdown Structure to include in the search. Select the **Include nested objects** option, if you want to include all objects under specified WBS items.
  11. Use the **Permission Group** tab to navigate the tree list for selecting the permission groups to include in the search.
  12. Use the **Object Type** tab to restrict the query to the specific object types with the list of disciplines.
  13. Use the **Volume** tab to restrict selection. Choose between two options for volume search. Select **Named Spaces** to designate the named spaces to include. Select **Planes** to specify certain reference planes or coordinate locations to define the six sides of a box. For objects contained in the volume inside this box, the software includes these objects in the filter.
  14. Use the **Properties** tab to restrict the search using properties of objects in the data model. For example, you can choose to match all properties listed in the grid, match any property listed in the grid, or use an operator, like equal ( = ) to narrow the search.

#### TIPS

- The **Select Properties** dialog box browses the data model to select properties on types. In the **Property** column, click **More**.
  - The **Select Object Type** dialog box specifies an object type for a property. You access this dialog box by clicking **More** in the **Object type** box on the **Select Properties** dialog box.
15. Use the **Reference 3D** tab (if available) to include any attached reference 3D models, folders or files in the query.
  16. Use the **Point Cloud** tab (if available) to include any point cloud objects in the query.
  17. Click **OK** on the **New Filter** dialog box to save the new filter and apply it to the selected objects in the workspace.

#### NOTES

- Use the **Configuration** tab to designate the options and configuration information for the filter. You can specify filter status and the associated permission group. These settings have no effect on the objects that the search returns. They govern the access permissions on the filter itself.
- An asking filter can have specific selection of filter definitions excluded, but have the **Asking Filter - user of filter will supply value** selected. You can specify these values when you

use the filter. For more information about creating an asking filter, see [Create a new asking filter](#).

- A compound filter combines two or more filters by using an operator, such as **not**, **union**, or **intersection**, between the filters to explain the relationship between the filters. For more information about creating a compound filter, see [Create a New Compound Filter](#).

## Model Properties Dialog Box

Sets Analysis Model options that are not available on the ribbon.

### See Also

*General Tab (Model Properties Dialog Box)* (on page 16)

*Relationship Tab* (on page 17)

*Configuration Tab* (on page 18)

*Notes Tab* (on page 19)

## General Tab (Model Properties Dialog Box)

### Category

Analysis Model properties are divided into different categories: Standard. You can select the category to define values for by using the **Category** option.

### Standard

#### Name

Specifies the name of the analysis model.

#### Name Rule

Select a name rule to use to name the analysis model. Select **User Defined** if you want to name the analysis model yourself.

#### Description

Type a description for the analysis model.

#### Frame Type

Select the frame type of the analysis model.

#### Analysis Method

Select the analysis method.

#### Analysis Assumptions

Type any assumptions made specific to the analysis of the model.

#### Export Date

Displays the date and time when the analysis model was last exported. This field is read-only. For more information on exporting analysis models, see [Export Analytical Models](#) (on page 25).



**Import Date**

Displays the date and time when the analysis model was last imported. This field is read-only. For more information on importing analysis and design results, see *Update from Analysis/Design* (on page 30).

**Coordinate System**

Select the coordinate system to use to transform the nodes locations in the analysis model data. You can create new coordinate systems in the **Grids** tasks.

**Selection Filter**

Select the selection filter used to assign members to the analysis model. Click **Create New Filter** to define a new filter. Click **More** to select an existing filter.

## Relationship Tab

Displays all objects related to the selected object for which you are viewing properties. For example, if you are viewing the properties of a pipe run, the related pipeline, features, parts, associated control points, hangers or supports, and equipment display on this tab. All WBS assignments, including project relationships, appear on this tab.

Additional examples for marine relationships are as follows:

- For plate and profile system properties, the related bounded objects, bounding objects, and connections are shown.
- For plate and profile system part properties, parent systems are shown.
- For assembly connection properties, all connected objects are shown.
- For the properties of a frame connection on a member, supported, supporting, and auxiliary supporting parts are shown.
- For split connection properties, the parent and auxiliary supporting parts are shown.

**Name**

Specifies the name of the object.

**Type**

Specifies the type of object. To change the options on the list, edit the **Weld Type** select list in Catalog.

**Go To**

Displays the properties of the selected object.

## Configuration Tab

Displays the creation, modification, and status information about an object.

 **NOTE** You cannot define the filters using the **Configuration** tab.

### Plant


Displays the name of the model. You cannot change this value.

### Permission Group

Specifies the permission group to which the object belongs. You can select another permission group, if needed. Permission groups are created in Project Management.


### Transfer

Reassigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This option is only available if the active model or project is replicated in a workshare configuration. The option is not available if all of the objects in the select set already belong to another location and are non-transferable. For more information, see *Transfer Ownership Dialog Box* in the *Common User's Guide*.

 **NOTE** The **Transfer** option does not apply to the filters and surface style rules.

### Approval State

Specifies the current status of the selected object or filter. The display depends on your access level. You might be unable to change the status of the object. The list is defined by the ApprovalStatus codelist.

 **NOTE** You can only edit or manipulate an object with a status of **Working**.

### Status

Specifies the location of the object in the workflow process. Changing this property sets the **Approval State**. The list is controlled by the ApprovalReason codelist in the ApprovalReason.xls file. You must bulkload this file. For more information, see *ApprovalReason* in the *Reference Data Guide*.

### Date Created

Specifies the creation date of the object.

### Created by

Specifies the name of the person who created the object.

### Date Last Modified

Specifies the date when the object was last modified.

### Last Modified by

Specifies the name of the person who last modified the object.

## ***Transfer Ownership Dialog Box***

Allows you to specify a new location and permission group for the selected model objects.

### **Current location**

Displays the name of the location with which the current permission group is associated. All of the objects in the select set must belong to the same location.

### **Current permission group**


Displays the name of the permission group with which the selected objects are currently associated. If all of the objects in the select set do not belong to the same permission group, this box appears blank.

### **New location**

Specifies the name of the location to which you want to assign the objects. In a global workshare configuration, this box lists all the locations in which you have write access to one or more permission groups. The selection in this box filters the entries in the **New permission group** box.


### **New permission group**

Specifies the new permission group to which to assign the selected objects. If you specify a value in the **New location** box, this list displays all permission groups to which you have write access in the selected location. If you do not specify a value in the **New location** box, this list includes all permission groups to which you have write access in all locations except the current location. This box is blank if you do not have write access to any permission groups at any locations other than the current one.

 **NOTE** We strongly recommend that administrators follow naming convention rules that include the location as a prefix in the permission group name.

## **Notes Tab**

Creates and edits user-definable text placed by the designer on an object in the model. The notes provide special instructions related to the object for the fabricator and are available in downstream tasks. For example, the notes appear in two-dimensional drawings and within design review sessions.

 **NOTE** Only one note of a given kind from a given object can be shown on a drawing. For example, if there are two fabrication notes on a piping part, then only one of the notes shows on the drawing. It is important to know about and to consider this situation when defining notes on an object in the modeling phase. For example, you can display one Fabrication note and one Installation note by defining two separate labels for the two kinds of notes.

### **Key point**

Specifies the key point on the object to which you want to add a note.

### **Notes at this location, listed by name**

Lists all notes for the selected key point on the object.

### **Date**

Displays the date that the note was created. The system automatically supplies the date.

**Time**

Displays the time that the note was created. The system automatically supplies the time.

**Purpose of note**

Specifies the purpose of the note.

**Author**

Displays the login name of the person who created the note. The system automatically supplies this information. You cannot change this information.

**Note text**

Defines the note text. The software does not limit the length of the note text.

**Show dimension**

Indicates that the note generates a dimension.

If you are displaying the properties for a Support component, then a dimension can be included for the component in the Support drawings, if you select the **Show dimension** option. The note must be associated with one of the key points for the Support component. It is recommended that you set the **Purpose of note** as **Fabrication**, but this is not a requirement. The note **Name** and **Note text** are not used when you select this option.

**New Note**

Creates a new note on the object.

**Standard Note**

Displays a list of standard notes from which you can select. This feature is not available in this version.

**Highlight Note**

Highlights the note in the graphic view so that you can easily find the note and the object to which it is related. This feature is not available in this version.

**Delete Note**

Deletes the currently displayed note.

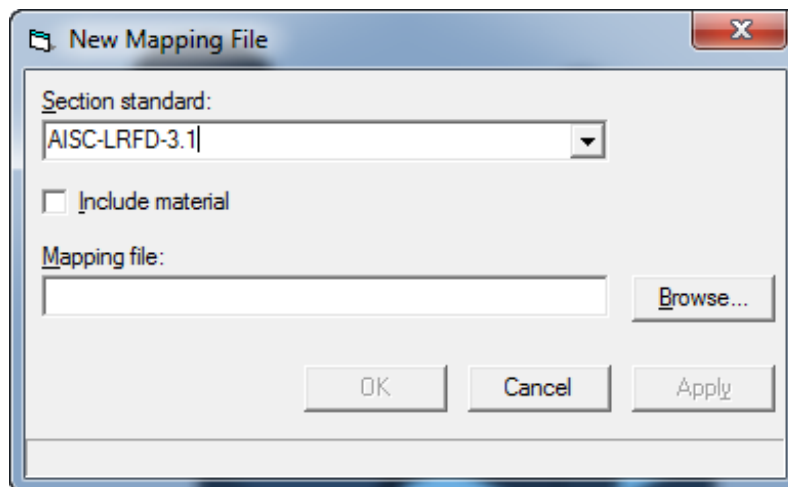
## SECTION 4

# New Mapping File

The **File > New Mapping File** command creates an XML mapping file for the section names, and optionally material names, used in the software and third-party application. Many times, the software and the third-party application use different names for the same section or material. The mapping file solves the naming conflicts by mapping section names in the software to section names in the other applications. The mapping file must contain each section standard table that you have used in the model.

**CAUTION** The mapping file created by this command is a template. The software does not write known-to-be-different section names to the mapping file. You are responsible for verifying, editing, updating, and maintaining the third-party application section names in the file.

### New Mapping File Dialog Box



#### Section standard

Select the section standard table for which to create a mapping file.

#### Include material

Select to include material name mappings in addition to the section name mappings.

#### Mapping file

Specify a name and folder path for the XML mapping file.

### Mapping File Format

The mapping file is an XML-formatted file with which you can define mappings for section names, material names, member types, slab types, and user-defined attribute/properties between Smart 3D and the third-party software. The format for each mapping is given below. You can include all five mappings in a single XML file, or you can define the mappings in separate XML files and reference the five files in a single master XML file using include

statements. Using separate files that are pulled together using include statements might be a better workflow as it allows you to quickly mix and match files for different requirements.

You cannot nest include files. Only the master XML mapping file can call an include file. You cannot call another include file inside an include file. An example of a master XML mapping file is shown below:

```
<xml>
<!-- Comment Line -->
  <IncludeXML href="SectionStandard.xml" />
  <IncludeXML href="Material.xml" />
  <IncludeXML href="MemberTypes.xml" />
  <IncludeXML href="SlabTypes.xml" />
  <IncludeXML href="UserAttribute_Map.xml" />
</xml>
```

## Section Mapping

The `<SectionStandard>` area maps the software section names to the third-party section names. The software section names are labeled **section name**. The third-party section names are labeled **externalname**. When the XML file is created, the software section name is duplicated for the third-party section name. You must verify that the correct third-party section name is defined for **externalname** by manually editing the XML file.

```
<xml>
<!-- Comment Line -->
<SectionStandard>
  <Standard name="AISC-LRFD-3.0" externalname="AISC" externalorganization="user"
    externaldate="2002" externalversion="1.0" />
    <Sections>
      <Section name="W10x39" externalname="W10x39" />
      <Section name="W10x33" externalname="W10x33" />
      <Section name="W10x30" externalname="W10x30" />
      ...
    </Sections>
  </SectionStandard>
</xml>
```

## Material Mapping

The `<MaterialStandard>` area maps the software material grade names to the third-party material grade names. You must have selected the **Include material** option when you created the XML file to see the material grade name mappings. The software materials are labeled **Material type** and **grade**. The third-party material names are labeled **externalname**. You must verify that the correct third-party material grade name is defined for **externalname** by manually editing the XML file. You must provide the external material standard name information by manually editing the XML file.

```
<xml>
<!-- Comment Line -->
<MaterialStandard>
  <Standard name="" externalname="" externalorganization="" externaldate="" externalversion="" />
    <Materials>
      <Material type="Steel - Carbon" grade="A36" externalname="A36" />
      <Material type="Steel - Carbon" grade="A529" externalname="A529" />
      <Material type="Steel - Carbon" grade="A588" externalname="A588" />
      ...
    </Materials>
  </MaterialStandard>
</xml>
```

## Member Type Mapping

The <MemberTypes> area maps the software member types to the third-party member types. You must create this section in a text editor. The software member types are labeled **Member type**. The third-party member types are labeled **externaltype** and **externalrole**.

```
<xml>
<!-- Comment Line -->
<MemberTypes>
  <Member type="Beam" externaltype="Beam" externalrole="" />
  <Member type="Girder" externaltype="Beam" externalrole="gantry_girder" />
  <Member type="Joist" externaltype="Beam" externalrole="joist" />
  ...
</MemberTypes>
</xml>
```

## Slab Type Mapping

The <SlabTypes> area maps the software slab types to the third-party slab types. You must create this section in a text editor. The software slab types are labeled **Slab type** and **composition**. The third-party slab types are labeled **externaltype**.

```
<xml>
<!-- Comment Line -->
<SlabTypes>
  <Slab type="4" Cast in Place" composition="CIP_4"_Fc3" externaltype="slab" />
  <Slab type="4" Cast in Place" composition="CIP_4"_Fc4" externaltype="flat_slab" />
  <Slab type="5" Cast in Place" composition="CIP_5"_Fc3" externaltype="wall" />
  ...
</SlabTypes>
</xml>
```

## User Attribute Mapping

The <UserAttributes> area maps third-party software attributes to Smart 3D properties. You must create this section in a text editor.

```
<xml>
<!-- Comment Line -->
<UserAttributes>
  <Object type="CSPSMemberSystemLinear" externaltype="assembly_design_structural_member_linear" role="" >
    <Interface name="IJUAStructuralFrameItemAttributes" externalname="IJUAStructuralFrameItemAttributes" >
      <Attribute name="item_number" externalname="item_number" />
      <Attribute name="item_name" externalname="item_name" />
      <Attribute name="item_description" externalname="item_description" />
      <Attribute name="life_cycle_stage" externalname="life_cycle_stage" />
    </Interface>
    <Interface name="IJUAStructuralFrameProductAttributes" externalname="IJUAStructuralFrameProductAttributes" >
      <Attribute name="life_cycle_stage" externalname="life_cycle_stage" />
    </Interface>

    <Interface name="IJUAAssemblyAttributes" externalname="IJUAAssemblyAttributes" >
      <Attribute name="assembly_sequence_number" externalname="assembly_sequence_number" />
      <Attribute name="complexity_level" externalname="complexity_level" />
    </Interface>

    <Interface name="IJUAADSMAttributes" externalname="IJUAADSMAttributes" >
      <Attribute name="key_member" externalname="key_member" />
      <Attribute name="structural_member_use" externalname="structural_member_use" />
      <Attribute name="Floor Thickness" externalname="Floor Thickness" />
      <Attribute name="structural_member_class" externalname="structural_member_class" />
    </Interface>
  </Object>
  ...
</UserAttributes>
</xml>
```

## Create a Mapping File

1. Click **File > New Mapping File**.
2. In the **Section standard** box, select the section standard for the mapping file.
3. Optionally, select **Include material** to write material names to the mapping file.
4. Click **Browse**, and then specify a name and folder location for the mapping file.
5. Click **OK**.
6. Edit the mapping file using a text editor such, as Notepad, and define the third-party standard section, material names, member types, and slab types.



## SECTION 5

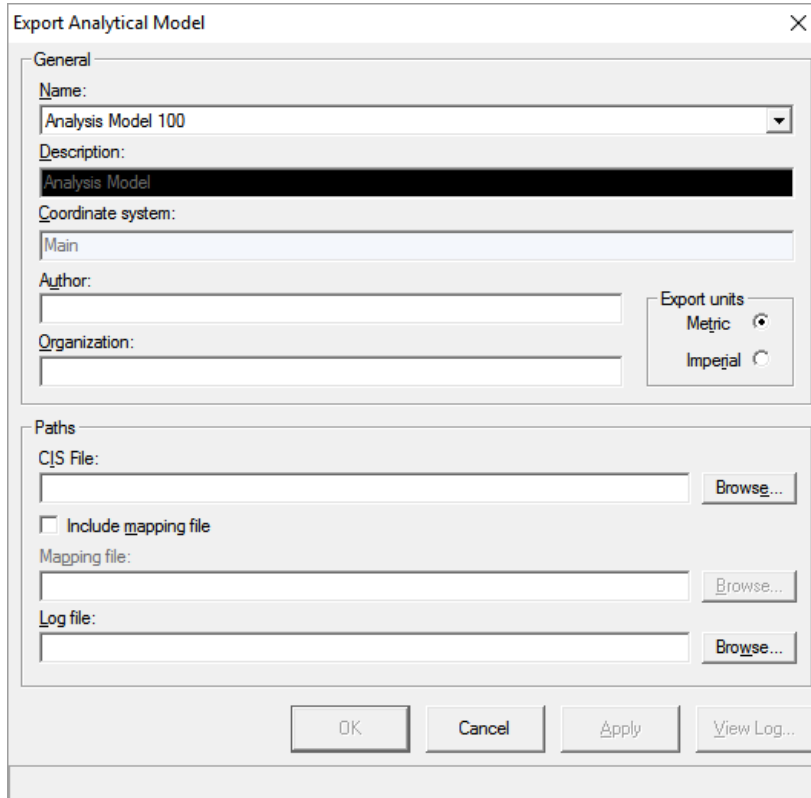
# Export Analytical Models

The **File > Export > Analytical Model** command writes a CIMsteel Integration Standard (CIS/2 format) file that contains all the necessary analytical data to allow a third party program to import the finite element representation. After being imported by the analytical program, the model is analyzed and designed to the appropriate section sizes, and then returned back to Smart 3D using the same file.

The CIS file contains all the necessary analytical data to allow third party programs to import the finite element representation of the analysis model. Generally, an analysis model is associated with a single CIS file. During the export process, nodes and finite elements are derived and written to the CIS files along with the boundary conditions, member releases, loads, and load cases and combinations. Because the nodes and finite elements only exist in the CIS file, the CIS file must be maintained throughout the life cycle of the analysis model.

If you need to update a third-party analysis model with changes made in the Smart 3D model after the initial analysis export from Smart 3D, you must use the same CIS file name for the update export. By using the same name, Smart 3D uses the existing file as input into the generation of the update export file. Smart 3D extracts the identity of the nodes and elements from the initial file and then uses this identification to create the new export file. If you do not use the same file name, Smart 3D creates new node and element identifications. Therefore, you will not be able to update the third-party analysis model because the node and element identifications will not match. For example, Model1A.stp was exported initially from Smart 3D. Now the Design Firm needs to send model changes to the Analysis Firm. The updated file to be exported must be named Model1A.stp. The original export file is renamed with a numbered prefix (based on the date) so the existing file is not over-written.

## Export Analytical Model Dialog Box



The dialog box is titled "Export Analytical Model". It contains two tabs: "General" and "Paths".

**General Tab:**


- Name:** A dropdown menu showing "Analysis Model 100".
- Description:** A text field showing "Analysis Model".
- Coordinate system:** A text field showing "Main".
- Author:** A text field.
- Organization:** A text field.
- Export units:** Two radio buttons, "Metric" (selected) and "Imperial".

**Paths Tab:**

- CIS File:** A text field with a "Browse..." button.
- Include mapping file:** A checkbox.
- Mapping file:** A text field with a "Browse..." button.
- Log file:** A text field with a "Browse..." button.

At the bottom of the dialog are four buttons: "OK", "Cancel", "Apply", and "View Log...".

### Name

Select the analysis model to export. You must create the analysis model using *New Analysis Models* (on page 11)  before you can export the analytical data that belongs to that analysis model using this command.

### Description

Displays the description included in the CIS file. The description is the one that you entered for the analysis model that you are exporting. This field is read-only. If you want to change the description, exit this command, and then edit the description found on the properties of the analysis model.

### Coordinate system

Displays the coordinate system to use to export the analysis model. The coordinate system is the one that you selected when you created the analysis model. This field is read-only. If you want to change the coordinate system, exit this command, and then change the coordinate system found on the properties of the analysis model. You can create new coordinate systems in the Grids task.

### Author

Specifies the person who created the CIS file. The default is the current system user name.

### Organization

Specifies your company or organization name to use in the CIS file.

### Export Units

Select the units of measure to use when writing the information to the CIS file.

### CIS file

Define the file name and folder path of the CIMsteel Integration Standard file to write. The default name is the name of the analysis model.

If you need to update a third-party analysis model with changes made in the Smart 3D model after the initial analysis export from Smart 3D, you must use the same CIS file name for the update export. By using the same name, Smart 3D uses the existing file as input into the generation of the update export file. Smart 3D extracts the identity of the nodes and elements from the initial file and then uses this identification to create the new export file. If you do not use the same file name, Smart 3D creates new node and element identifications. Therefore, you will not be able to update the third-party analysis model because the node and element identifications will not match. For example, Model1A.stp was exported initially from Smart 3D. Now the Design Firm needs to send model changes to the Analysis Firm. The updated file to be exported must be named Model1A.stp. The original export file is renamed with a numbered prefix (based on the date) so the existing file is not over-written.

### Include mapping file

Select this option to use a section name mapping file when exporting the member to the CIS file. A mapping file swaps the Smart 3D name for a section (for example, L3-1/2X2-1/2X1/4) with the third-party analysis software name for a section (for example, L3.5X2.5X1/4). You must create the mapping file using the File > New Mapping File before you can use the mapping file in this command.

### Mapping file

Specify the mapping file to use if the **Include mapping file** option is selected.

### Log file

Specify a log file name. You can view the log file by clicking **View Log**.

### NOTES

- Only loads belonging to a load case that is part of a load combination definition are written to the CIS file.
- The software does not write curved members to the CIS file.

If you have any questions about using this translator, please contact Intergraph support. You can find support information on our web site <http://support.intergraph.com>.

### Based-on CIS/2 Statement

<b>Application Name:</b>	Intergraph Smart™ 3D
<b>Application Version:</b> Version 2016 (11.0)	<b>Date:</b> Monday, September 12, 2016
<b>Translator Version:</b> Version 2016 (11.0)	<b>Date:</b> Monday, September 12, 2016
<b>Software Vendor:</b>	Intergraph 305 Intergraph Way Madison, Alabama 35758 U.S.A.

The translators for this application have been implemented in accordance with the second release of the CIMsteel Integration Standards (CIS/2.0) for the following (combination of) Conformance Classes:

- CC110, CC166, CC170, CC177, CC221, CC216, CC100, CC331, CC307

**Type of CIS Translator:** Basic | **DMC** | IDI | PMR-enabled

**Data exchange capabilities:** Import | **Export** | Import & Export

**Level of implementation:** **File Exchange** | In memory | DBMS | KBS

**Flavors supported:** EU | **US** | UK

**Unit Systems supported:** **SI** | US Imperial

The vendor places the following riders on the operation of the translators:

- element\_eccentricity is not exported.
- managed\_data\_deleted, managed\_data\_creation, and managed\_data\_transaction are not exported.

**Date of Statement:** Monday, September 12, 2016

**Statement made by:** Intergraph

## Export analysis model


1. Click **File > Export > Analytical Model**.
2. Select the analysis model to export.
3. Type your name in the **Author** box and your company name in the **Organization** box.
4. Specify the units of measure to use for the CIS file.
5. Specify the file name and folder for the CIS file.

★ **IMPORTANT** If you need to update a third-party analysis model with changes made in the Smart 3D model after the initial analysis export from Smart 3D, you must use the same CIS file name for the update export. By using the same name, Smart 3D uses the existing file as input into the generation of the update export file. Smart 3D extracts the identity of the nodes and elements from the initial file and then uses this identification to create the new export file. If you do not use the same file name, Smart 3D creates new node and element identifications. Therefore, you will not be able to update the third-party analysis model because the node and element identifications will not match. For example, Model1A.stp was exported initially from Smart 3D. Now the Design Firm needs to send model changes to the Analysis Firm. The updated file to be exported must be named Model1A.stp. The original export file is renamed with a numbered prefix (based on the date) so the existing file is not over-written.

6. Define a mapping file, if needed.
7. Define a log file name and folder.
8. Click **Apply**.


9. When processing is complete, click **View Log** to see the log file.
10. Click **Cancel** to exit the dialog box.

### **NOTES**

- Only loads belonging to a load case that is part of a load combination definition are written to the CIS file.
- You must create an analysis model using the *New Analysis Models* (on page 11)  before you can export the analysis model.
- You must create a mapping file using the File > New Mapping File before you can use that mapping file when exporting the analysis model.

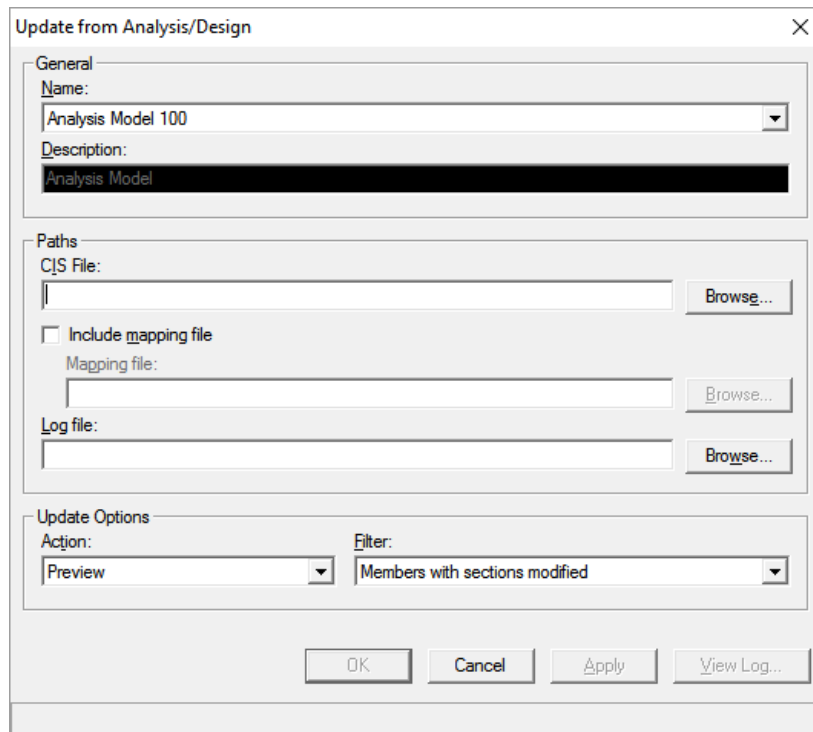
## SECTION 6

# Update from Analysis/Design


 Opens the **Update from Analysis/Design** dialog box, which is used to read a CIMsteel Integration Standard (CIS/2 format) file that contains all the analysis and design results from a third party program. This command reads members section sizes only. Member topological changes, new loads, new load cases, and new load combinations are not read from the CIS file.

★ **IMPORTANT** The active permission group must have write access to all members whose sections must be updated. If the active permission group does not have write access to all the members, then none of the members are updated.

### Update from Analysis/Design Dialog Box



#### Name

Select the Analysis Model for which to import results. You must export the Analysis Model to a CIS file using *Export Analytical Models* (on page 25)  before you can import the analysis and design results using this command.

#### Description

Displays the description for the Analysis Model that was included in the CIS file. This box is read-only.

**CIS file**

Browse to and select the CIMsteel Integration Standard file to read.

**Include mapping file**

Select this option to use a section name mapping file when importing the member section sizes from the CIS file. A mapping file swaps the third-party analysis software name for a section (for example, L3.5X2.5X1/4) with the Smart 3D name for a section (for example, L3-1/2X2-1/2X1/4). You must create the mapping file using the File > New Mapping File before you can use the mapping file in this command.

**Mapping file**

Specify the mapping file to use if the **Mapping file** option is selected.

**Log file**

Specify a log file name. You can view the log file by clicking **Review Log**.

**Action**

Specify whether to import the member section sizes from the CIS file or just preview the changes.

**Filter**

Select a filter to use to query against the information in the CIS file. Use this option with the **Action** option to compare the contents of the CIS to the Analysis Model members. The available filters are:

- **Members with sections modified** - Highlights or updates (depending on the **Action** setting) members in the model whose section size has changed in the CIS file.
- **Members with sections not modified** - Highlights members in the model whose section size has not changed in the CIS file. This option is available only when **Action** is set to **Preview**.
- **Members with sections not found in catalog** - Writes section sizes that are in the CIS file but that are not available in the Catalog to the log file. This option is available only when **Action** is set to **Preview**.
- **Members in analytical model & in CIS file** - Highlights members in the model that are found in the CIS file. This option is available only when **Action** is set to **Preview**.
- **Members in analytical model & not in CIS file** - Highlights members in the model that are not found in the CIS file. This option is available only when **Action** is set to **Preview**.

**Status**

Displays the results of the update process and the overall status of the Analysis Model.

If you have any questions about using this translator, please contact Intergraph support. You can find support information on our web site <http://support.intergraph.com>.

**Based-on CIS/2 Statement**

**Application Name:**

Smart 3D

**Application Version:** Version 2016 (11.0)

**Date:** Monday, September 12, 2016

**Translator Version:** Version 2016 (11.0)

**Date:** Monday, September 12, 2016

**Software Vendor:**

Intergraph Process, Power & Marine  
305 Intergraph Way  
Madison, Alabama 35758 U.S.A.

The translators for this application have been implemented in accordance with the second release of the CIMsteel Integration Standards (CIS/2.0) for the following (combination of) Conformance Classes:

- CC110 + CC170 + CC177 + CC331 + CC100

**Type of CIS Translator:** Basic | **DMC** | IDI | PMR-enabled

**Data exchange capabilities:** **Import** | Export | Import & Export

**Level of implementation:** **File Exchange** | In memory | DBMS | KBS

**Flavors supported:** EU | **US** | UK

**Unit Systems supported:** **SI** | US Imperial

The vendor places the following riders on the operation of the translators:

- The import updates sections of existing members only. The import does not create any new objects within the application.

**Date of Statement:** Monday, September 12, 2016

**Statement made by:** Intergraph Process, Power & Marine


---

### What do you want to do?


- *Preview analysis/design results* (on page [33](#))
  - *Update analysis model using analysis/design results* (on page [33](#))
-



## Preview analysis/design results

1. Click **Update from Analysis/Design**  on the vertical toolbar.
2. Specify the CIS file to read.
3. Specify the mapping file information if you are using one.
4. Set the **Action** option to **Preview**.
5. Click **Apply**.

## Update analysis model using analysis/design results

1. Click **Update from Analysis/Design**  on the vertical toolbar.
2. Specify the location of the CIS to read.
3. Specify the mapping file information if you are using one.
4. In the **Action** option, select **Update**.
5. Click **OK**.

## SECTION 7

# Loads

### Load Cases and Load Combinations

You define load cases interactively using the software. Load case types are those defined in the CIS/2 specification.

In addition to load cases, you can define, edit, and delete load combinations interactively. Load combinations are associated with an analysis model. Therefore, you need to create an analysis model before defining any load combinations.

### Placing Loads

As you place a load, you must first choose the type of load to place. The software gives you three options: concentrated, distributed, and partially distributed. Then you can select the load case to apply. You can define a magnitude at each end of a partially distributed load, which allows for trapezoidal loading. You can also give the direction of a load (Force X, Force Y, Force Z, Moment RX, Moment RY, and Moment RZ), as well as specify the magnitude of the load itself. Additionally, you can place multiple loads on a member; however, you cannot place loads on curved members.

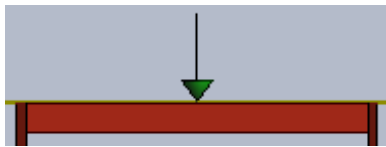
After loads are placed, you can always come back and edit them, changing any of their attributes, except load type. You can also delete loads. In cases where members have multiple loads, you can delete as many of them as you like.

The three types of loads are distributed, concentrated, and partially distributed:

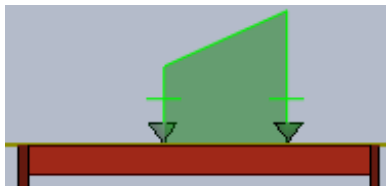
- A distributed load is uniformly distributed along the member length.



- A concentrated load is applied at any user-defined location along the member's length.

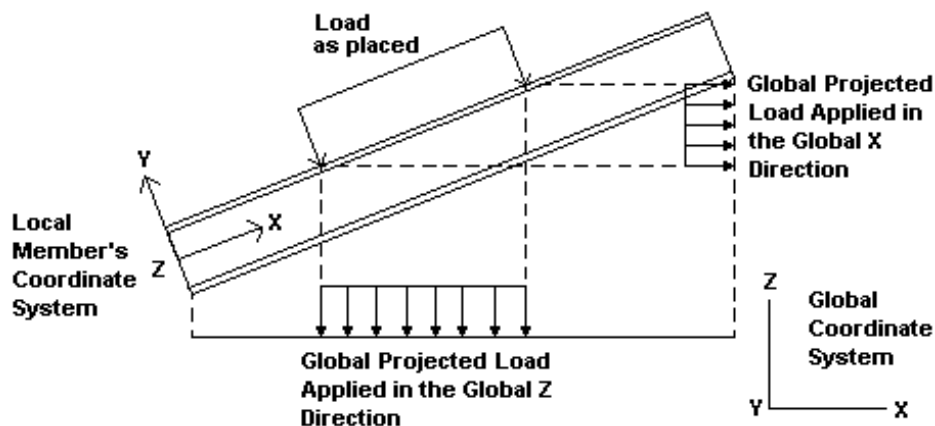


- A partially distributed load is applied between user-specified start and end points anywhere along the member's length.

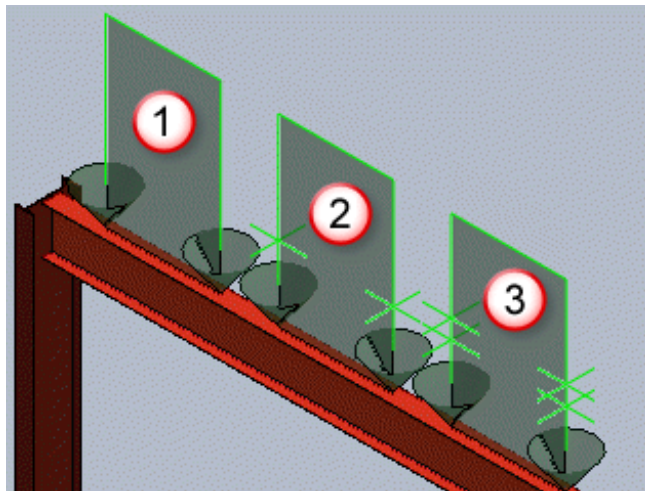


The directions of loads, the load reference, can be defined in three ways:

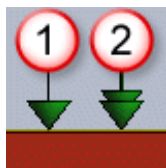
- A Local load uses the member's local coordinate system to define the load direction.
- A Global load is applied to the member in the direction of the global coordinate axis specified.
- A Project Global load is applied to the projection of the member on the plane perpendicular to the global direction specified and acting in the direction specified.




You can distinguish the load's reference by the number of x symbols on the load graphic. If there is no x, the load reference is local (1); one x is global (2); and two x symbols is global projected (3).



Loads can be placed in the X, Y, Z, RX, RY, or RZ direction. Force loads placed in X, Y, and Z directions have a single arrowhead (1). Moment loads, placed in the RX, RY, and RZ directions, have a double arrowhead (2).



## Loads and Member Splitting

If after you have placed loads on a member you use the **Place Split**  command in the Structure task, the software recalculates loads as follows:

- Concentrated loads stay where they were placed (the physical location along the original member part length). The absolute or relative placement value is recalculated based on the new member part length.
- Distributed loads are split into two distributed loads with the same magnitude (one distributed load for each new member part).
- Partially distributed loads do one of two things based on the split location. If the split location is between the partially distributed load's end points, the load is split into two partially distributed loads one on each new member part on either side of the split location.

If the split location is outside of the partially distributed load's end points, the load is unaffected other than having the end points' absolute or relative placement values recalculated based on the new member part length.


---

## What do you want to do?

- *New Load Cases* (on page [37](#))
  - *New Load Combinations* (on page 40)
  - *Place New Concentrated Loads* (on page 45)
  - *Place New Distributed Loads* (on page 55)
-

## SECTION 8

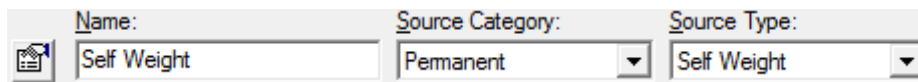
# New Load Cases

 Creates load cases. Load cases are used to group loads based on their source. For example, all self-weight loads can be grouped into one load case and all wind loads can be grouped into another load case. For analysis, load cases are combined into a load combination to give a combined load effect.

Load cases are stored in the model. To store load cases in the Catalog database, you must use the **Copy to Catalog** command.

### Load Case Ribbon

Displays options for modifying load cases.



The screenshot shows a ribbon interface with three main sections: 'Name:', 'Source Category:', and 'Source Type:'. Each section has a text input field and a dropdown arrow. The 'Name:' field contains 'Self Weight'. The 'Source Category:' dropdown is set to 'Permanent'. The 'Source Type:' dropdown is set to 'Self Weight'.

### Properties

Opens the **Load Case Properties** dialog box that you can use to set additional load case properties that are not available on the ribbon. For more information, see *Load Case Properties Dialog Box* (on page 39).

#### Name

Specifies the name for the load case.

#### Source category

Specifies the physical action source and duration of the load case.

#### Source type

Specifies the source of the physical action for the load case.

### General Tab (Load Case Properties Dialog Box)

#### Category

Load Case properties are divided into different categories: Standard. You can select the category to define values for by using the **Category** option.

#### Standard

##### Name

Specifies the name of the load case.

##### Name Rule

Select a name rule to use to name the load case.

- Select **Default Name Rule** to have the load case named using the following syntax: LoadCase-workshare location ID-sequence number. For example: LoadCase-1-0001, where 1 is the workshare location ID and 0001 is the sequence number.

- Select **User Defined** if you want to name the load case yourself.

**Source category**

Specifies the physical action source and duration of the load case.

**Source type**


Specifies the source of the physical action for the load case.

---


**What do you want to do?**

- *Create a load case* (on page 38)
  - *Delete a load case* (on page 38)
  - *Add a load case to a load combination* (on page 39)
- 

## Create a load case

1. Click **New Load Case**  on the vertical toolbar.
2. Type a name for the load case.
3. Select a source category for the load case.
4. Select a source type for the load case.
5. Click **OK**.


## Delete a load case

1. Click **Select**  on the vertical toolbar.
2. Select **Load Cases** in the **Locate Filter**.
3. Select the **Analysis** tab on the **Workspace Explorer**.
4. Right-click on the load case to delete, and then select **Delete**.

### **CAUTIONS**

- If you delete a load case, all loads placed using that load case are put on the To Do List as in error until you define a new load case for them.
- If you delete a load case that is used in a load combination, the load combination is automatically edited to remove the deleted load case.

## Add a load case to a load combination

1. Click **Select**  on the vertical toolbar.
2. Select **Load Combinations** in the **Locate Filter**.
3. Select the **Analysis** tab in the **Workspace Explorer**.
4. Expand the **Load Combinations** folder under the analysis model.
5. Right-click on the load combination, and then select **Properties**.
6. Select the **Load Cases** tab.
7. Add the load case to the last row.
8. Click **OK**.

## Load Case Properties Dialog Box

Sets load case options that are not available on the ribbon.

### See Also

*General Tab (Load Case Properties Dialog Box)* (on page 39)

*General Tab (Load Combination Properties Dialog Box)* (on page 43)

*Relationship Tab* (on page 17)

*Configuration Tab* (on page 18)

*Notes Tab* (on page 19)

## General Tab (Load Case Properties Dialog Box)

### Category

Load Case properties are divided into different categories: Standard. You can select the category to define values for by using the **Category** option.

### Standard

#### Name

Specifies the name of the load case.

#### Name Rule

Select a name rule to use to name the load case.

- Select **Default Name Rule** to have the load case named using the following syntax:  
LoadCase-workshare location ID-sequence number. For example: LoadCase-1-0001,  
where 1 is the workshare location ID and 0001 is the sequence number.
- Select **User Defined** if you want to name the load case yourself.

#### Source category



Specifies the physical action source and duration of the load case.

#### Source type

Specifies the source of the physical action for the load case.

## SECTION 9

# New Load Combinations

 Creates, edits, and deletes load combinations. You should have at least one load case defined before you define load combinations. For more information on creating load cases, see *New Load Cases* (on page 37) .

★ **IMPORTANT** You must have an analysis model defined before you can create load combinations. For more information, see *New Analysis Models* (on page 11).

### Load Combination Ribbon

Displays options for modifying load combinations.



The screenshot shows a software interface with two input fields. The first field is labeled 'Analysis Model:' and contains a dropdown menu with 'Analysis Model 1' selected. The second field is labeled 'Name:' and contains the text 'Load Comb 1'.

### Properties

Opens the **Load Combination Properties** dialog box that you can use to set additional load combination properties that are not available on the ribbon. For more information, see *Load Combination Properties Dialog Box* (on page 43).

### Analysis Model

Select the analysis model in which to create the load combination.

### Name

Specifies the name for the load combination.

### General Tab (Load Combination Dialog Box)

#### Category

Load combination properties are divided into different categories: Standard. You can select the category to define values for by using the **Category** option.

#### Standard

##### Name

Specifies the name of the load combination.

##### Name Rule

Select a name rule to use to name the load combination.

- Select **Default Name Rule** to have the load combination named using the following syntax: LoadCombination-workshare location ID-sequence number. For example: LoadCombination-1-0001, where 1 is the workshare location ID and 0001 is the sequence number.
- Select **User Defined** if you want to name the load combination yourself in the **Name** field.



**Analysis Model**

Specifies the analysis model to place the load combination in.

**Load Cases Tab (Load Combination Dialog Box)****Name**

Displays the name of the load combination.

**Load Case**

Select the load case to add to the load combination.


**Factor**

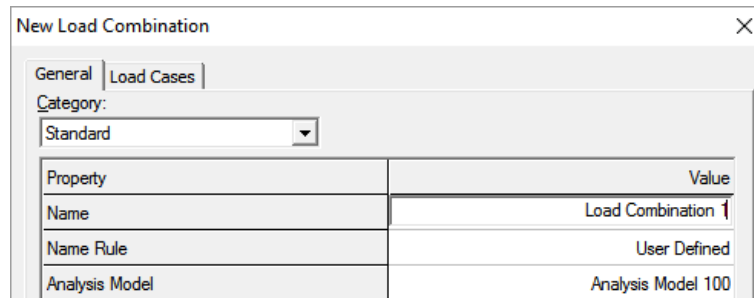
Enter the multiplier for the load case.

**What do you want to do?**

- *Create a load combination* (on page 41)
- *Add a load case to a load combination* (on page 39)
- *Remove a load case from a load combination* (on page 42)
- *Delete a load combination* (on page 43)

## Create a load combination

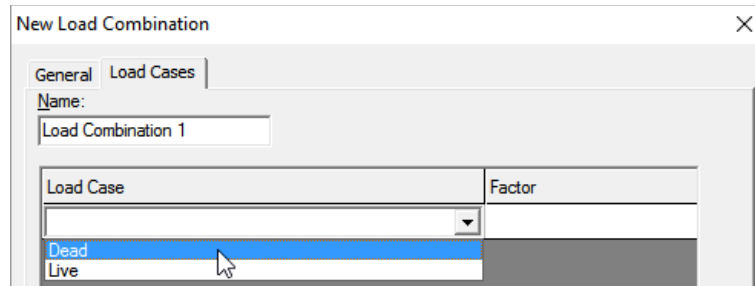
1. Click **New Load Combination**  on the vertical toolbar.
2. In the **Name** box, type a name for the load combination.



Property	Value
Name	Load Combination 1
Name Rule	User Defined
Analysis Model	Analysis Model 100

3. Select the **Load Cases** tab.
4. Click inside the first row under **Load Case**.

5. Select a load case to add to the load combination.



6. Type a factor for the load case.
7. Continue adding load cases as necessary.
8. Click **OK**.


## Add a load case to a load combination

1. Click **Select** on the vertical toolbar.
2. Select **Load Combinations** in the **Locate Filter**.
3. Select the **Analysis** tab in the **Workspace Explorer**.
4. Expand the **Load Combinations** folder under the analysis model.
5. Right-click on the load combination, and then select **Properties**.
6. Select the **Load Cases** tab.
7. Add the load case to the last row.
8. Click **OK**.

## Remove a load case from a load combination

1. Click **Select** on the vertical toolbar.
2. Select **Load Combinations** in the **Locate Filter**.
3. Select the **Analysis** tab in the **Workspace Explorer**.
4. Expand the **Load Combinations** folder under the analysis model.
5. Right-click on the load combination, and then select **Properties**.
6. Select the **Load Cases** tab.
7. Select the load case to remove.
8. Click **Remove**.
9. Click **OK**.

## Delete a load combination

1. Click **Select**  on the vertical toolbar.
2. Select **Load Combinations** in the **Locate Filter**.
3. Select the **Analysis** tab in the **Workspace Explorer**.
4. Expand the **Load Combinations** folder under the analysis model.
5. Right-click on the load combination to delete, and then select **Delete**.

## Load Combination Properties Dialog Box

Sets load combination options that are not available on the ribbon.

### See Also

*General Tab (Load Combination Properties Dialog Box)* (on page 43)

*Load Cases Tab (Load Combination Properties Dialog Box)* (on page 44)

*Relationship Tab* (on page 17)

*Configuration Tab* (on page 18)

*Notes Tab* (on page 19)

## General Tab (Load Combination Properties Dialog Box)

### Category

Load combination properties are divided into different categories: Standard. You can select the category to define values for by using the **Category** option.

### Standard

#### Name

Specifies the name of the load combination.

#### Name Rule

Select a name rule to use to name the load combination.

- Select **Default Name Rule** to have the load combination named using the following syntax: LoadCombination-workshare location ID-sequence number. For example: LoadCombination-1-0001, where 1 is the workshare location ID and 0001 is the sequence number.
- Select **User Defined** if you want to name the load combination yourself in the **Name** field.

#### Analysis Model

Specifies the analysis model to place the load combination in.

## **Load Cases Tab (Load Combination Properties Dialog Box)**

### **Name**

Displays the name of the load combination.

### **Load Case**

Select the load case to add to the load combination.

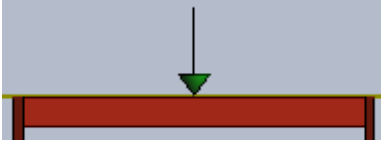
### **Factor**

Enter the multiplier for the load case.

## SECTION 10

# Place New Concentrated Loads

↓ Places a load that acts on a small area of a member. Concentrated loads represent loads from other members, the weight of equipment, or maybe piping resting on the member.



### NOTES

- Only loads belonging to a load case that is part of a load combination definition are written to the CIS file.
- You cannot place loads on curved members.

### Place New Concentrated Load Ribbon

Sets options for placing a concentrated load.

Load Case: **Dead** Load: **Force X** Reference: **Local**

Position As: **Relative** Position: **0** Magnitude: **1.00 kip** Name:

### Properties

Opens the **Concentrated Load Properties** dialog box that you can use to set load point options not available on the ribbon. For more information, see *Concentrated Load Properties Dialog Box* (on page 53).

### Select Member

Select the members on which to place the concentrated load.

### Select Position

Identify the location of the concentrated load along the member. You can use the **Tools > PinPoint** and **Tools > Point Along** commands to help locate the concentrated load on the member.

### Finish

Places the concentrated load on the member.

### Cancel

Clears the current selection set.

### Accept

Accepts the current settings and activates the **Finish** button.

### Load Case

Select the load case to which the concentrated load should be assigned. Create new load cases by using *New Load Cases* (on page 37) .

### Load

Select the direction in which the load magnitude is to be applied. You can place loads in any of the six directions: Force X, Force Y, Force Z, Moment RX, Moment RY, or Moment RZ.

### Reference

Select the coordinate system reference for the load. For more information, see *Loads* (on page 34).

### Position As

Specifies how the load location is calculated from the start of the member. The location can be defined as **Relative** or **Absolute**.

A relative location can be thought of as a distance percentage along the member. For example, a load placed at a relative distance of 0.333 is located at one-third the member length measured from the member start. A load placed at a relative distance of 0.5 is located at the middle of the member.

An absolute distance is the actual distance from the start of the member. The software verifies that the absolute distance that you specify is not past the end of the member.

### Position

Specify the position of the load.

### Magnitude

Type the force of the load. The magnitude can be a positive or negative value, as needed, and is applied in the direction that you selected in the **Load** box.

### Name


Type a name for the concentrated load.

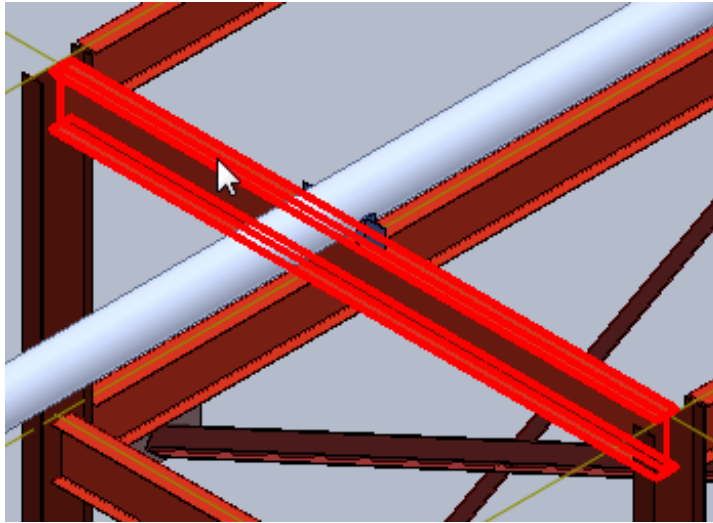
---

### What do you want to do?

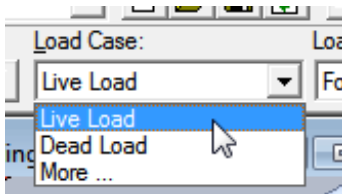
- *Place concentrated load by absolute key-in* (on page 47)
  - *Place concentrated load by relative key-in* (on page 48)
  - *Place concentrated load using point along* (on page 50)
  - *Modify concentrated load magnitude* (on page 52)
  - *Move a concentrated load* (on page 52)
  - *Delete a concentrated load* (on page 53)
-

## Place concentrated load by absolute key-in


1. Click **Place New Concentrated Load** .
2. Select the members on which to place the load.

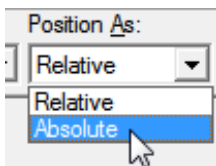


3. Click **Accept** .
4. Select the load case for the load. Use **New Load Cases** (on page 37) to create new load cases.

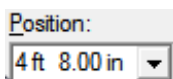


5. Set the **Load** and **Reference** options.
 

 **TIP** For more information on the **Load** and **Reference** options, see *Loads* (on page 34).
6. Type a value in the **Magnitude** box.
7. Set the **Position As** option to **Absolute**.

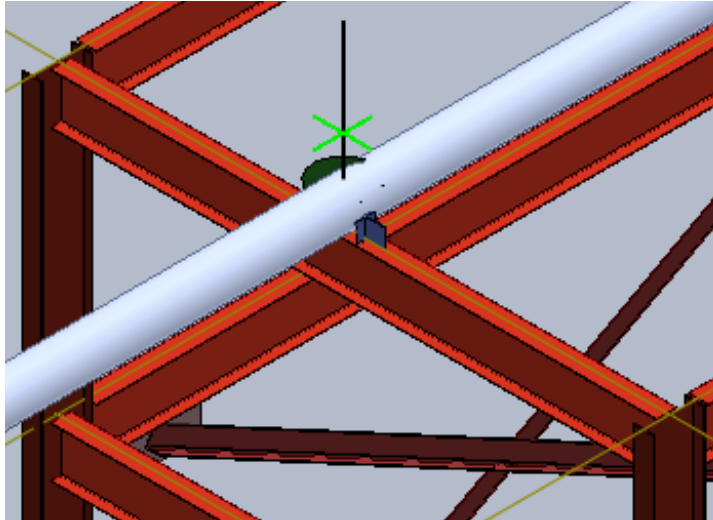


8. In the **Position** box, type the absolute distance from the start of the member where you want the load.



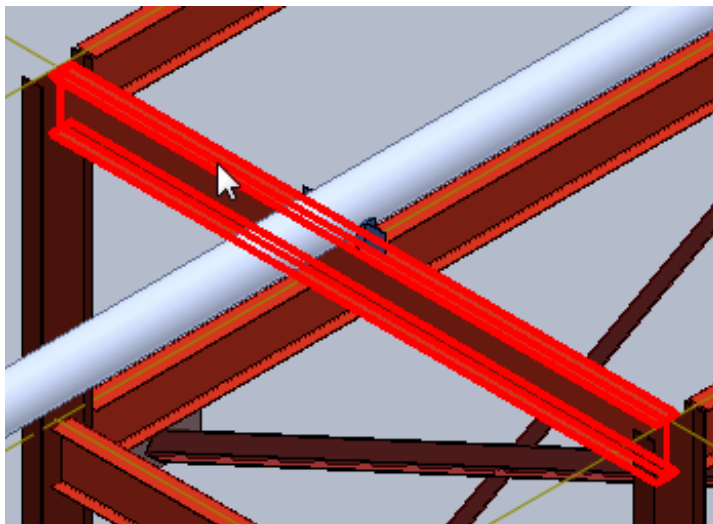
9. Click **Accept** .


10. Click **Finish**.



## Place concentrated load by relative key-in

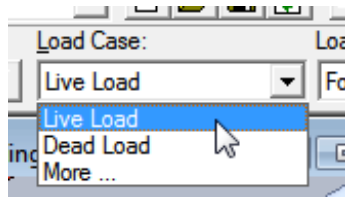
1. Click **Place New Concentrated Load** .



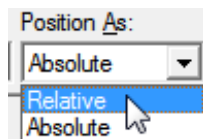
2. Select the members on which to place the load.
3. Click **Accept** .



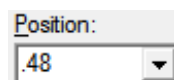
4. Select the load case for the load. Use **New Load Cases** (on page 37) to create new load cases.



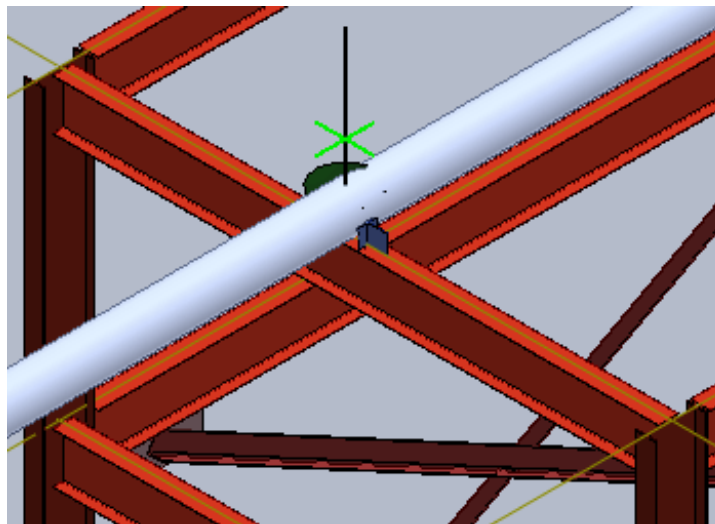
5. Set the **Load** and **Reference** options. For more information on the **Load** and **Reference** options, see *Loads* (on page 34).
6. Type a value in the **Magnitude** box.
7. Set the **Position As** option to **Relative**.



8. In the **Position** box, type the relative position of the load from the start of the member. Valid values are 0.0 to 1.0, with 0.0 being the start of the member and 1.0 being the end of the member. For example, type **0.5** if you want the load at the middle of the member, or type **0.333** if you want the load a third of the member length from the start of the member.

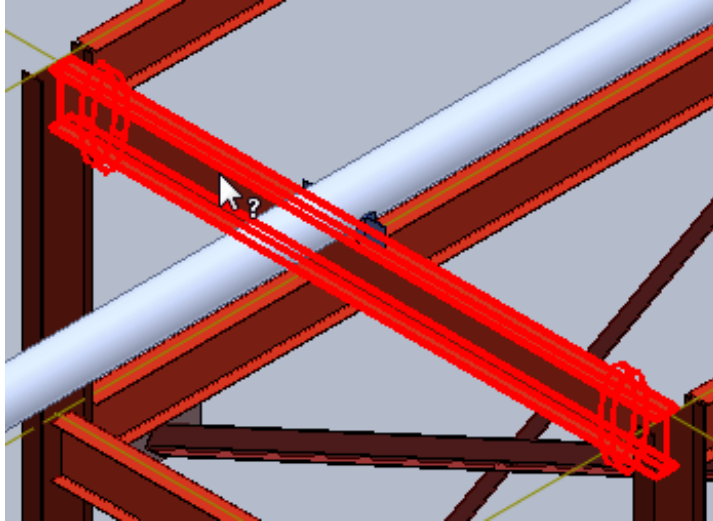


9. Click **Accept** ✓.
10. Click **Finish**.

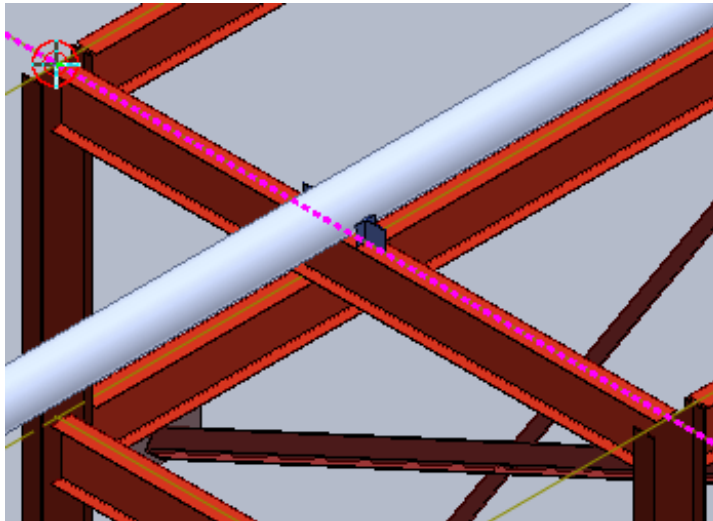


## Place concentrated load using point along

1. Click **Tools > Point Along** to activate the **Point Along** command.
2. Select the member on which the load is going to be placed as the object to measure along.

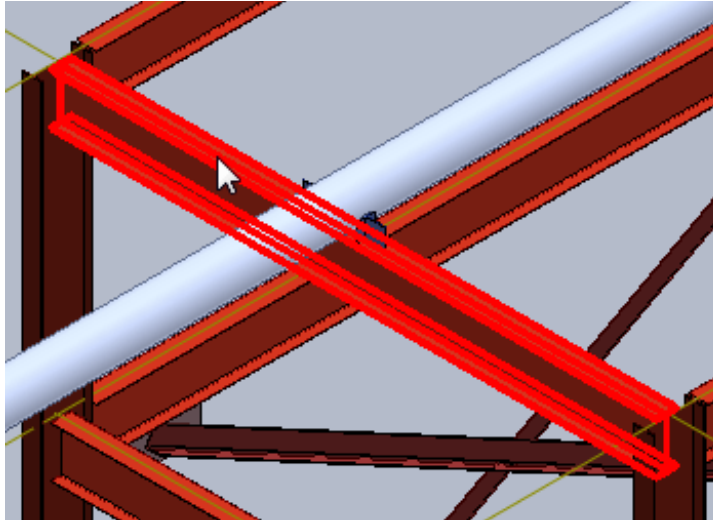


3. Place the reference point at the start of the member.

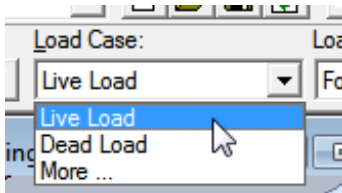


4. Select **Place New Concentrated Load** .

5. Select the member on which to place the load.

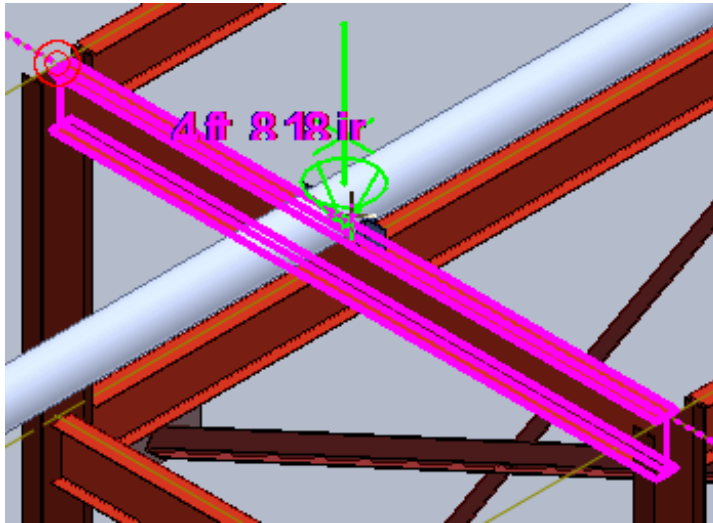


6. Click **Accept** ✓.
7. Select the load case for the load. Use **New Load Cases** (on page 37) to create new load cases.




8. Set the **Load** and **Reference** options. For more information on the **Load** and **Reference** options, see *Loads* (on page 34).
9. Type a value in the **Magnitude** box.

10. Identify the location of the load on the member by either dragging the cursor along the member until the distance you want displays or by typing a distance in the **Distance** box on the Point Along ribbon.




11. Click **Accept** ✓.
12. Click **Finish**.



## Modify concentrated load magnitude

1. Click **Select**  on the vertical toolbar.
2. Select **Loads** in the **Locate Filter**.
3. Select the load to modify.
4. Select the **Magnitude** box, and then type a new force for the load.

## Move a concentrated load

1. Click **Select**  on the vertical toolbar.
2. Select **Loads** in the **Locate Filter**.
3. Select the load to move.
4. Select the **Position** box, and then type a new location for the load relative to the start of the member.

## Delete a concentrated load

1. Click **Select**  on the vertical toolbar.
2. Select **Loads** in the **Locate Filter**.
3. Select the load to delete.
4. Click **Delete** .

## Concentrated Load Properties Dialog Box

Sets concentrated load options that are not available on the ribbon.

### See Also

*General Tab (Concentrated Load Properties Dialog Box)* (on page 53)

*Relationship Tab* (on page 17)

*Configuration Tab* (on page 18)

*Notes Tab* (on page 19)

## General Tab (Concentrated Load Properties Dialog Box)

### Category

Concentrated load properties are divided into different categories: Standard. You can select the category to define values for by using the **Category** option.

### Standard

#### Name

Specifies the name of the concentrated load.

#### Name Rule

Select a name rule to use to name the concentrated load.

- Select **Default Name Rule** to use the following naming syntax: Concentrated<Load>-<Workshare Location ID>-<Sequence Number>. For example, ConcentratedForce-1-0002, where Concentrated is hard coded text, Force is the <Load> (either Force or Moment), 1 is the <Workshare Location ID>, and 0002 is the <Sequence Number>, in this case, the second Concentrated Force load placed.
- Select **User Defined** if you want to name the load yourself in the **Name** field.

#### Analysis Parent

Displays the name load's parent object.

#### Load Case

Select the load case to which the concentrated load should be assigned. Create new load cases by using *New Load Cases* (on page 37).

#### Load

Select the direction in which the load magnitude is to be applied. You can place loads in any of the six directions: Force X, Force Y, Force Z, Moment RX, Moment RY, or Moment RZ.

### Reference

Select the coordinate system reference for the load. For more information, see *Loads* (on page 34).

### Position As

Specifies how the load location is calculated from the start of the member. The location can be defined as **Relative** or **Absolute**.

A relative location can be thought of as a distance percentage along the member. For example, a load placed at a relative distance of 0.333 is located at one-third the member length measured from the member start. A load placed at a relative distance of 0.5 is located at the middle of the member.

An absolute distance is the actual distance from the start of the member. The software verifies that the absolute distance that you specify is not past the end of the member.

### Position


Specify the position of the load.

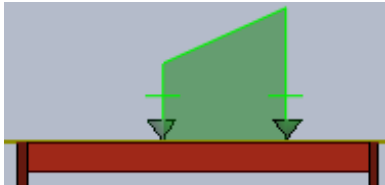
### Magnitude

Type the force of the load. The magnitude can be a positive or negative value, as needed, and is applied in the direction that you selected in the **Load** box.

## SECTION 11

# Place New Distributed Loads

 Places a load distributed along the full or partial length of a member. The load source could be a wall resting on the member, a slab supported by the member, or perhaps wind pressure.

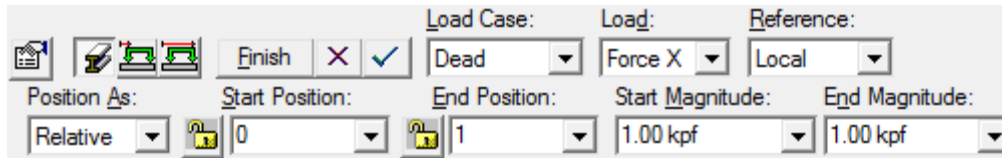


### NOTES

- Only loads belonging to a load case that is part of a load combination definition are written to the CIS file.
- You cannot place loads on curved members.

### Place New Distributed Load Ribbon

Sets options for placing a distributed load.



Load Case:		Load:		Reference:	
Dead		Force X		Local	
Position As:	Start Position:	End Position:	Start Magnitude:	End Magnitude:	
Relative	0	1	1.00 kpf	1.00 kpf	

### Properties

Opens the **Distributed Load Properties** dialog box that you can use to set load options not available on the ribbon. For more information, see *Distributed Load Properties Dialog Box* (on page 61).

### Select Member

Select the members on which to place the distributed load.

### Select start position

Specify the start location of the distributed load on the member.

### Select end position

Specify the end location of the distributed load on the member.

### Finish

Places the distributed load on the member.


### Cancel

Clears the current selection set.

**✓ Accept**

Accepts the current settings and activates the **Finish** button.

**Load Case**

Select the load case to which the distributed load should be assigned. Create new load cases by using *New Load Cases* (on page 37) .

**Load**

Select the direction in which the load magnitude is to be applied. You can place loads in any of the six directions: X, Y, Z, RX, RY, or RZ.

**Reference**

Select the load frame. For more information, see *Loads* (on page 34).

**Position As**

Specifies how the load location is calculated from the start of the member. The location can be defined as **Relative** or **Absolute**.

A relative location can be thought of as a distance percentage along the member. For example, a load placed at a relative distance of 0.333 is located at one-third the member length measured from the member start. A load placed at a relative distance of 0.5 is located at the middle of the member.

An absolute distance is the actual distance from the start of the member. The software verifies that the absolute distance that you specify is not past the end of the member.

**Start Position**

Identify the start location of the distributed load along the member. You can use the **Tools > PinPoint** and **Tools > Point Along** commands to help locate the distributed load on the member.

**End Position**

Identify the ending location of the distributed load along the member. You can use the **Tools > PinPoint** and **Tools > Point Along** commands to help locate the distributed load on the member.

**Start Magnitude**

Type the force of the load at the starting location. The magnitude can be a positive or negative value, as needed, and is applied in the direction that you selected in the **Load** box.

**End Magnitude**

Type the force of the load at the ending location. The magnitude can be a positive or negative value, as needed, and is applied in the direction that you selected in the **Load** box.


---

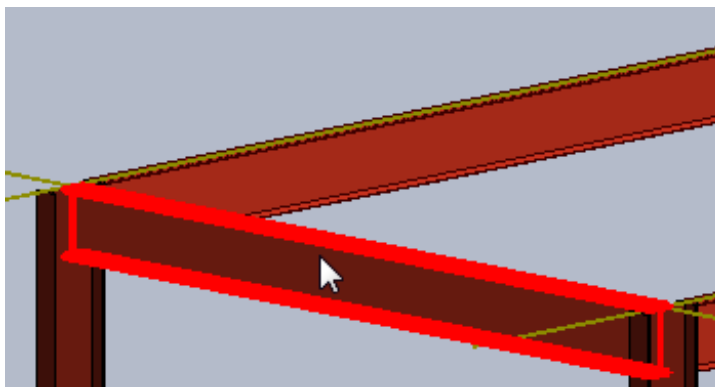
**What do you want to do?**


- *Place a fully distributed load* (on page 57)
  - *Place a partially distributed load* (on page 58)
  - *Modify distributed load magnitude* (on page 59)
  - *Delete a distributed load* (on page 61)
-

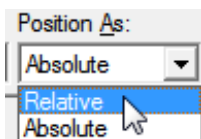




## Place a fully distributed load

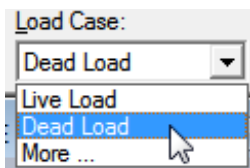
1. Click **Place New Distributed Load** .
2. Select the members on which to place the load.



3. Click **Accept** .
4. In the **Position As** option, select **Relative**.

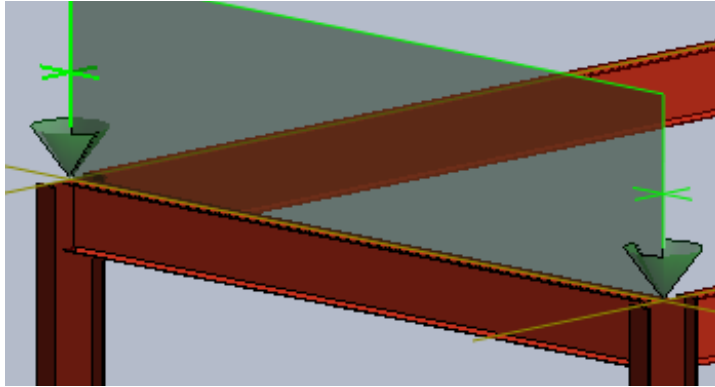


5. In the **Start Position** box, type 0.
6. Click **Accept** .
7. In the **End Position** box, type 1.
8. Click **Accept** .
9. Select a load case for the distributed load. Use *New Load Cases* (on page 37) to create new load cases.




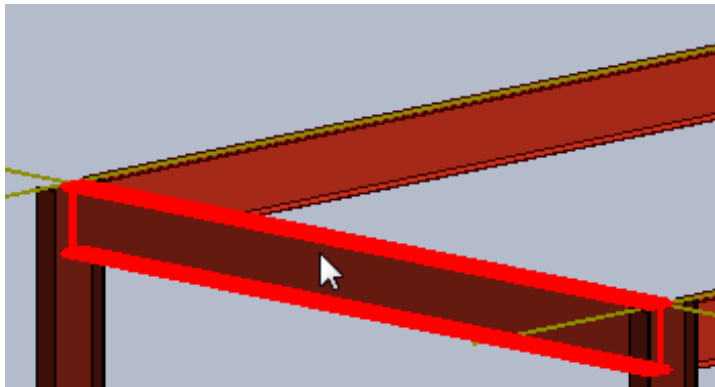
10. Specify options for the **Load** and **Reference** options. For more information on the **Load** and **Reference** options, see *Loads* (on page 34).
11. Define the forces in the **Start magnitude** and **End magnitude** boxes.


- Click **Finish**.

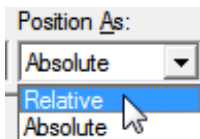




## Place a partially distributed load

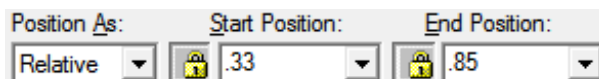
- Click **Place New Distributed Load** .
- Select the members on which to place the load.



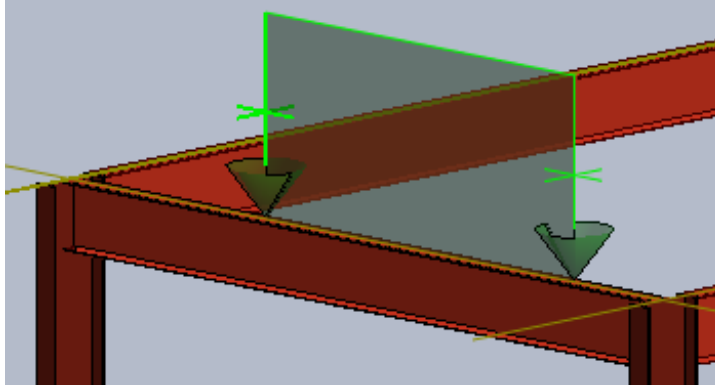
- Click **Accept** .
- In the **Position As** option, select **Relative**.




- In the **Start Position** box, define the start location of the load.
- Click **Accept** .
- In the **End Position** box, define the end location of the load.
- Click **Accept** .

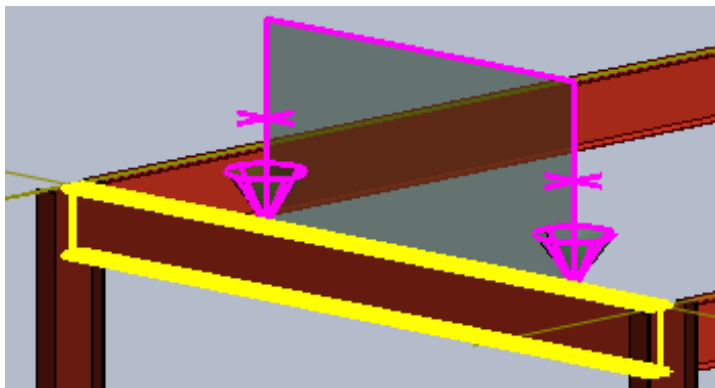


9. Select a load case for the distributed load. Use *New Load Cases* (on page 37) to create new load cases.
10. Specify options for the **Load** and **Reference** options. For more information on the **Load** and **Reference** options, see *Loads* (on page 34).
11. Define the forces in the **Start magnitude** and **End magnitude** boxes.
12. Click **Finish**.



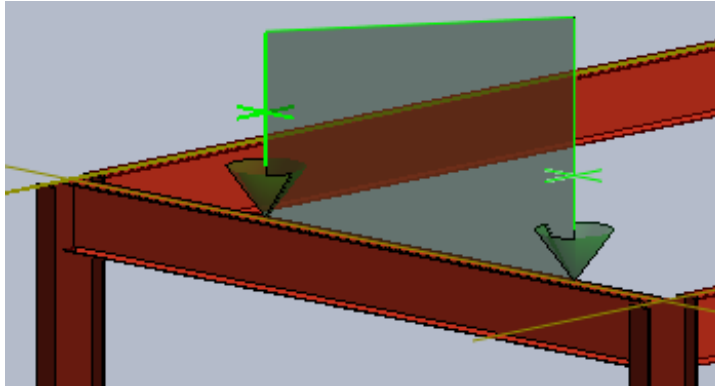
## Modify distributed load magnitude

1. Click **Select**  on the vertical toolbar.
2. Select **Loads** in the **Locate Filter**.
3. Select the load to modify.



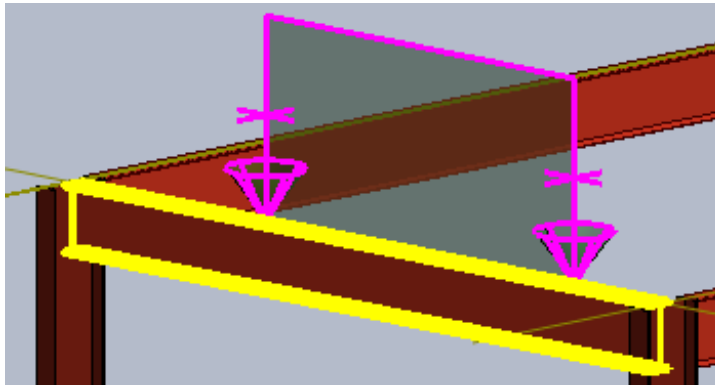
4. Select the **Start Magnitude** box, and then type a new force.

5. Select the **End Magnitude** box, and then type a new force.

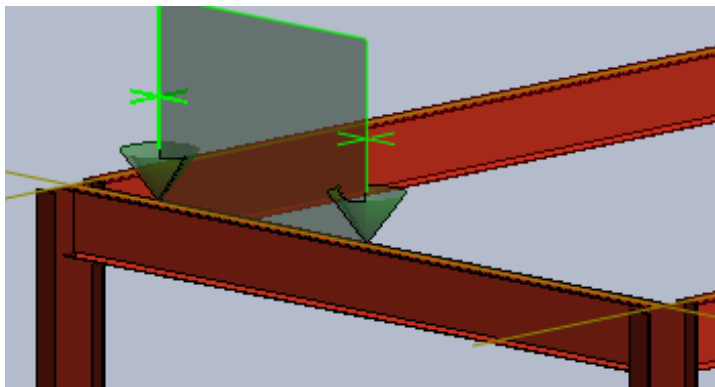


## Modify distributed load position



1. Click **Select** on the vertical toolbar.
2. Select **Loads** in the **Locate Filter**.
3. Select the load to modify.



4. Select the **Start Position** box, and then type a new starting location.
5. Select the **End Position** box, and then type a new ending location.



## Delete a distributed load

1. Click **Select**  on the vertical toolbar.
2. Select **Loads** in the **Locate Filter**.
3. Select the load to delete.
4. Click **Delete** .

## Distributed Load Properties Dialog Box

Sets distributed load options that are not available on the ribbon.

### See Also

*General Tab (Distributed Load Properties Dialog Box)* (on page 61)

*Relationship Tab* (on page 17)

*Configuration Tab* (on page 18)

*Notes Tab* (on page 19)

## General Tab (Distributed Load Properties Dialog Box)

### Category

Distributed load properties are divided into different categories: Standard. You can select the category to define values for by using the **Category** option.

### Standard

#### Name

Specifies the name of the load.

#### Name Rule


Select a name rule to use to name the load.

- Select **Default Name Rule** to use the following naming syntax: Distributed<Load>-<Workshare Location ID>-<Sequence Number>. For example, DistributedForce-1-0002, where Distributed is hard coded text, Force is the <Load> (either Force or Moment), 1 is the <Workshare Location ID>, and 0002 is the <Sequence Number>, in this case, the second Distributed Force load placed.
- Select **User Defined** if you want to name the load yourself in the **Name** field.

#### Analysis Parent

Displays the name load's parent object.

#### Load Case

Select the load case to which the load should be assigned. Create new load cases by using *New Load Cases* (on page 37) .

#### Load

Select the direction in which the load magnitude is to be applied. You can place loads in any of the six directions: Force X, Force Y, Force Z, Moment RX, Moment RY, or Moment RZ.

### Reference

Select the coordinate system reference for the load. For more information, see *Loads* (on page 34).

### Position As

Specifies how the load location is calculated from the start of the member. The location can be defined as **Relative** or **Absolute**.

A relative location can be thought of as a distance percentage along the member. For example, a load placed at a relative distance of 0.333 is located at one-third the member length measured from the member start. A load placed at a relative distance of 0.5 is located at the middle of the member.

An absolute distance is the actual distance from the start of the member. The software verifies that the absolute distance that you specify is not past the end of the member.

### Start Position

Specify the start position of the load.

### End Position

Specify the end position of the load.

### Start Magnitude


Type the force of the load. The magnitude can be a positive or negative value, as needed, and is applied in the direction that you selected in the **Direction** box.

### End Magnitude

Type the force of the load. The magnitude can be a positive or negative value, as needed, and is applied in the direction that you selected in the **Direction** box.

## SECTION 12

# Set Boundary Conditions

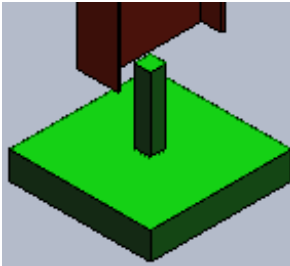
 Defines the support conditions that occur between the end of a member and the ground. All boundary conditions' degrees of freedom (DOF) are written to the CIS/2 file relative to the coordinate system defined for the analysis model to which the boundary condition is assigned.

★ **IMPORTANT** You must define an analysis model before you can define boundary conditions. For more information, see *New Analysis Models* (on page 11).

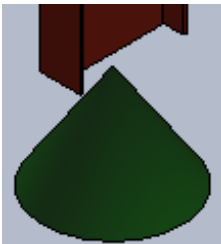
### Supports

Supports, also known as boundary conditions, define how a structure is restrained from rigid body motion under the influence of external loads. Supports are defined using the six degrees of freedom; X, Y, Z, RX, RY, and RZ; applied in the global coordinate system.


This software displays supports using a unique indicator that appears at the member end. Indicators for fully fixed and RX, RY, and RZ fixed supports look like this:



Indicators for supports where RX, RY, RZ, or a combination of the three moment directions are released look like this:



You can pause over the support for a tool tip that displays the fixed/released directions of that support.

When you split a member system (using **Place Split**  in the Structure task) that has boundary conditions defined, the boundary conditions remain on the ends for which they were defined. For example, if end 1 of the original member was fully supported (X, Y, Z, RX, RY, RZ) and end 2 was supported only in RX, and RZ. Then end 1 of new member 1 gets fully supported and end 2 of new member 2 gets the RX and RZ support. The new interior ends (end 2 for member 1 and end 1 for member 2) do not have any supports defined.


## Releases

Releases define the type of connection; rigid, released, or elastic; between two member part ends. Releases are defined using the six degrees of freedom: X, Y, Z, RX, RY, and RZ; applied in the local coordinate system of the member part.

A rigid connection does not allow the member part end to move in the specified direction at the connection.

A released connection does allow the member part end to move freely in the specified direction at the connection.

Releases are defined using the **End Releases** category on the **Member Part** tab of the **Member Part Prismatic Properties** dialog box.

When you split a member system (using **Place Split**  in the Structure task) that has end releases defined for one of its member parts, the member end releases of the original member part are copied to the two new member parts.

## Set Boundary Condition Ribbon

Contains options for placing boundary conditions in the model.



### Properties

Opens the **Boundary Conditions Properties** dialog box. Use this dialog box to define boundary condition properties that are not available on the ribbon. For more information, see *Boundary Condition Properties Dialog Box* (on page 66).

### Select Frame Connections

Select the frame connection where you would like to apply a boundary condition. You can select one or more frame connections at a time.

### Finish

Places the boundary conditions in the model.

### Select

Filters the frame connections that you can place. Select **All** to select all frame connections. Select **Without boundary conditions** to select only those frame connections that do not have a boundary condition defined.

### Cancel

Reject the current selection.

### Accept

Accept the current selection.

### Analysis Model

Displays the name of the analysis model to which the boundary conditions are assigned.

### Type

Select the type of boundary condition to place.



**Name**



Type a name for the boundary condition. If you do not type a name, the software automatically assigns a name based on a naming rule.

---



**What do you want to do?**

- *Place fixed boundary condition* (on page 65)
  - *Place pinned boundary condition* (on page 65)
  - *Modify a boundary condition* (on page 65)
  - *Removing a boundary condition* (on page 66)
  - *Define end releases* (on page 66)
- 


## Place fixed boundary condition

1. Click **Set Boundary Condition** .
2. Select the frame connection at the end of the member at which to place the boundary condition.
3. In the **Type** option, select **Fix: All**.
4. Click **Accept** .
5. Click **Finish**.



## Place pinned boundary condition

1. Click **Set Boundary Condition**  on the vertical toolbar.
2. Select the frame connection at the end of the member at which to place the boundary condition.
3. In the **Type** option, select the pinned option to place.
4. Click **Accept** .
5. Click **Finish**.


## Modify a boundary condition

1. Click **Select**  on the vertical toolbar.
2. Select **Boundary Conditions** in the **Locate Filter**.
3. Select the boundary condition to modify.
4. Modify the boundary condition as needed.

## Removing a boundary condition

1. Click **Select**  on the vertical toolbar.
2. Select **Boundary Conditions** in the **Locate Filter**.
3. Select the boundary condition to remove.
4. Click **Delete** .

## Define end releases

1. Click **Select**  on the vertical toolbar.
2. Select **Member Parts** in the **Locate Filter**.
3. Right-click on the member part, and then select **Properties**.
4. Select the **Member Part** tab.
5. In the **Category** option, select **End Releases**.
6. Specify the end releases as required.
7. Click **OK**.

## Boundary Condition Properties Dialog Box

Sets boundary condition options that are not available on the ribbon.

### See Also

*General Tab (Boundary Condition Properties Dialog Box)* (on page 67)

*Relationship Tab* (on page 17)

*Configuration Tab* (on page 18)

*Notes Tab* (on page 19)

## General Tab (Boundary Condition Properties Dialog Box)

### Category

Boundary conditions (supports) properties are divided into different categories: Standard. You can select the category to define values for by using the **Category** option.

### Standard

#### Name

Specifies the name of the boundary condition.

#### Name Rule

Select a name rule to use to name the boundary condition.

- Select **Default Name Rule** to use the following naming syntax: BoundaryCondition-<Workshare Location ID>-<Sequence Number>. For example, BoundaryCondition-1-0002, where BoundaryCondition is hard coded text, 1 is the <Workshare Location ID>, and 0002 is the <Sequence Number>, in this case, the second boundary condition placed.
- Select **User Defined** if you want to name the boundary condition yourself in the **Name** field.

#### Analysis Model

Displays the name of the analysis model to which the boundary condition belongs.

#### Boundary Condition Type

Specify the type of boundary condition to place. Select **User Defined** to fix and release individual directions yourself by using the appropriate box.

#### X Displacement

Specifies if this direction is free or fixed.

#### Y Displacement

Specifies if this direction is free or fixed.

#### Z Displacement

Specifies if this direction is free or fixed.

#### X Rotation

Specifies if this direction is free or fixed.

#### Y Rotation

Specifies if this direction is free or fixed.

#### Z Rotation

Specifies if this direction is free or fixed.

# Glossary

## ***ACI***

American Concrete Institute.

## ***Active Template Library (ATL)***

Set of class templates and wizards supplied with Microsoft C++ Version 5.0 and later. You can use an ATL when you create ActiveX controls and any other type of object that uses the Component Object Model (COM) model. Using an ATL is generally preferred over Microsoft Foundation Classes (MFC), because the implementations are smaller, easier to use, and more closely tied to the COM model.

## ***AISC (American Institute of Steel Construction)***

An organization responsible for defining American steel construction standards.

## ***AISI***

American Iron and Steel Institute

## ***analysis***

The process of modeling a structure to study its physical behavior, such as mechanical (static and dynamic), thermal, and so forth. The most commonly-used analysis is finite element.

## ***analytical member***

A mathematical object derived from the logical member used to perform finite element analysis and design.

## ***approval state***

Recorded state of acceptance of information contained in objects within the database. The approval states indicate a level of confidence in the information stored in the database and govern your ability to alter specific data about a product.

## ***attribute***

A single type of non-graphics information that is stored about an object such as diameter or end preparation.

## ***axis***

An imaginary line used to define the orientation of a system or object normally defined in terms of an x-, y-, and z-axis. Some 3-D graphic objects have an associated axis used to define the center or axis for rotations.

## ***bay***

The distance between two trusses.

## ***BCSA (British Constructional Steelwork Association)***

An organization responsible for defining British steel construction standards.

***beam***

A structural member type typically placed with the member axis in a nominal horizontal orientation.

***bearing plate***

A steel plate used to distribute a load over a larger area. Usually used at the base of a column.

***bill of material (BOM)***

Hierarchical decomposition of a product into constituent assemblies and parts. Specific types of BOMs exist (for example, an EBOM is a bill of material from the point of view of an engineering department; an MBOM is a bill of material from the point of view of manufacturing).

***boundary condition***

A property that defines the restriction on the allowable direction of movement (degree of freedom) at a particular node.

***bulkload***

The process by which reference data in Microsoft Excel workbooks is loaded into the Catalog database.

***catalog***

Repository of information about components and materials used in construction. When you use catalog parts in the model, the software places an occurrence of the catalog part in the project. This occurrence is a copy of the actual catalog part.

***Catalog database***

The database that contains the reference data. Each model database can reference a different Catalog database.

***CISC (Canadian Institute of Steel Construction)***

An organization responsible for defining Canadian steel construction standards.

***constraint***

An analytical boundary condition applied to a node in the model. Constraints can be applied in any valid degree of freedom in the model. Constraints force zero movement at the node and degree of freedom of application.

***constraints***

A logical restriction that controls how part symbols ports relate to each other and to reference ports. There are four constraints: parallel, perpendicular, coincident, and distance.

***contract***

A Work Breakdown Structure object representing a scope of work, usually performed by an external supplier. The contract is related to a project and appears in the Work Breakdown Structure hierarchy.

***coordinate***

The location of a point along the X-, Y-, or Z-axis.

***coordinate system***

A geometric relation used to denote the location of points in the model. The most common coordinate system is the rectangular coordinate system, whereby points are located by traversing the X-, Y-, and Z-axes of the model. Normally, coordinate systems have their origin defined as 0,0,0.

***cutting plane***

A plane that cuts through an object.

***degree of freedom***

An allowable direction of movement, either translation or rotation. There are six possible degrees of freedom (DOFs): translation X, Y, and Z, and rotation RX, RY, and RZ.

***drawing tool***

Tool that helps in the process of creating, modifying, or manipulating objects. Examples are PinPoint and SmartSketch.

***easting***

A term that describes an east coordinate location in a coordinate system.

***end releases***

Physical member properties that define the connection between a member and its nodes. End releases (degrees of freedom) are used to simulate pinned members as well as other special modeling situations.

The member will not contribute stiffness to the node if the degree of freedom (end release) is released. Similarly, the node will not transfer forces or moments to the member through a release degree of freedom. End releases can be defined in any valid degree of freedom for the model.

***fabricate***

To cut, punch, and sub-assemble members in the shop.

***fasteners***

Bolts and rivets used to connect structural members.

***element***

Primitive geometric shape such as a line, circle, or arc.

***field adjustment***

Material added to the neat design geometry of piping or structural parts to allow for fit up in the case that extra material is required due to uncontrolled variance in the manufacturing and construction process.

***finite element***

A simple geometric shape defined by a specific number of nodes in a specific order. Elements are dependent on all the nodes defining their shape; if any node is deleted, the element is also deleted. Elements are the building blocks of finite element models. Elements can be one of three types: linear or one-dimensional, plate or two-dimensional, or solid or three-dimensional.

***flavor***

A different variation of a symbol. Each variation has different occurrence property values.

***focus of rotation***

A point or line about which an object or view turns.

***full penetration weld***

A type of weld in which the weld material extends through the complete thickness of the components being joined.

***generic specific***

Object that is parametrically defined or defined to suit a family of specific parts (for example, International Standards parametrics). For example, a 100 - 200 gpm pump in the catalog can provide a general shape to appear in the model until a specific object has been identified. See also specific and specific object.

***GUIDs***

Acronym that stands for Globally Unique Identifiers. The software automatically creates the GUIDs sheet in the Excel workbooks when you create the Catalog database and schema. The purpose of storing GUIDs within Excel workbooks is to help you keep track of what has been loaded into the database. Storing GUIDs also helps to avoid the situation in which a replacement Catalog database causes existing models to become invalid.

***initial structural plan***

Principal structural plan for the plant; also called a construction profile.

***interference checking***

A process that identifies possible collisions or insufficient clearance between objects in the model.

***kinematics analysis***

Analysis of mechanical motion.

***kips***

Kilo pounds.

***ksi***

Kips per square inch.

***leg length analysis***

Preferred term is welding length analysis.

***lintel***

A horizontal member used to carry a wall over an opening.

***load (structure)***

A force vector applied to a member.

***load group***

A grouping in which all components feature uniform load limits and stress safety characteristics. For example, if a pipe clamp from load group 5 has a maximum nominal load of 20kN, then so does a threaded rod from load group 5.

***logical member***

An object in the model used to represent the design topology.

***material analysis***

Analysis of a completed design work for extracting detailed material requirements; also called material lists.

***material list***

An option category that controls the format and content of the bill of materials.

***material properties***

Properties of the material useful in the analysis process.

***member name***

A user-definable alphanumeric code used to uniquely identify individual members in the model.

***member part***

A model object derived from the logical model that represents the manufactured physical member parts.

***member system***

A logical collection of member parts that can be moved as a single entity.

***move from point***

Starting point for an action. For example, when you move an equipment object, the Move From point determines the point of origin for the move.

***move to point***

Ending point for an action. For example, when you move an equipment object, the Move To point determines where you want the move to stop.

***MTO neutral file***

A non-graphic output file that can be fed into a material control system. MTO stands for Material Take-Off.

***node***

- One of the set of discrete points in a flow graph.
- A terminal of any branch of a network or a terminal common to two or more branches of a network.
- An end point of any branch or a network or graph, or a junction common to two or more branches.



***northing***

A term that describes a north coordinate location in a coordinate system.

***object***

A type of data other than the native graphic format of the application.

***occurrence (of part or equipment)***

Instantiation of a part of equipment in the model that refers to the part library; an instance of a specific object. The design can be built several times, and therefore the occurrence can apply to more than one hull. Typically, an occurrence points back to a specific object, either for its complete definition, as in the case of a particular valve, or for its made from material, as in the case of a steel plate part cut from sheets. Thus, when a designer selects a component from the catalog and places it at a location in the space of the plant, the software creates an occurrence of that object in the plant design.

***occurrence property***

A characteristic that applies to an individual object in the model. Occurrence properties are designated with 'oa.' in the reference data workbooks. You can view and modify occurrence properties on the Occurrence tab of the properties dialog boxes in the software. Depending on the object, some occurrence properties are read-only.

***origin***

In coordinate geometry, the point where the X-, Y-, and Z-axes intersect.

***origin point***

The point at which the coordinate system is placed, providing a full Cartesian coordinate system with positive and negative quadrants. Points are placed at coordinates relative to the origin point, represented by the X, Y, and Z values.

***orthogonal***

The characteristic of an element consisting completely of elements positioned at 90-degree angles. A square is an orthogonal element.

***orthographic***

A depiction of an object created by projecting its features onto a plane along lines perpendicular to the plane.

***orthotropic material***

A material that has two material directions that are orthogonal to one another. An example of an orthotropic material is wood.

***principle of superposition***

The principle that states that the stresses, strains, and displacements due to different forces can be combined. This principle is only valid for linear analysis.

***reference data***

The data that is necessary to design plants or ships using the software. Reference data includes graphical information, such as symbols. It also contains tabular information, such as physical dimensions and piping specifications.

***route***

1) A line connecting a series of points in space and constituting a proposed or traveled route. 2) The set of links and junctions joined in series to establish a connection.

***stress***

Forces acting on structural members due to various types of loads. These forces can be shear, tension, compression, or torsion.

***structure analysis***

Analysis routines that provide stress and deflection data for structural designs. Loading conditions can be both static and dynamic. Finite element analysis is the most common type of structure analysis.

***stud***

A bolt, threaded on both ends, used to connect components.

***suspended floor***

A concrete floor system built above and off the ground.

***symmetric node***

Type of vertex on a curve. A curve with a symmetric node has the same curvature on each side of the node. A handle can be attached to a symmetric node for editing.

***system***

A conceptual design grouping that organizes parts in hierarchical relationships. A system represents a functional view of the model and includes information such as system name, type, properties, and design specifications for the objects assigned to the system.

***target point***

The origin for coordinate measurements displayed by PinPoint. You can position the target point anywhere on the drawing sheet or view.

***tolerant geometry***

A type of ACIS geometry - either an edge or a vertex - that is outside the tolerance for ACIS and requires special handling.

***trimmed surface***

A surface whose boundary is fully or partially inside the "natural" geometric definition of the surface. Some or the entire control polygon extends outside the face boundary.

***trunk***

Feature that quickly reserves space for the distributive systems and other systems that have a path. Along the trunk are stations that define the cross section and identify part or system membership.

***unit/module modeler***

Facility of the system to structure collections of equipment and components into a single identifiable object.

***user attributes***

A customized property in the reference data. The Custom Interfaces sheets in the Excel workbooks define these properties. You can list the customized properties on the individual part class sheets.

***vertex***

A topological object that represents a point in the three-dimensional model.

***weight and CG analysis***

Routines that compute the weight of commodity materials as configured in a given design (for example, plate and pipe) and determine total weight and center of gravity (CG) for a collection of material and equipment, as well as the complete plant.

***welding***

Weld requirements for joining materials. Welding length analysis is the calculation of required weld dimensions; also called leg length analysis.

***working plane***

The available 2-D plane of movement for endpoint selection.

***workset***

Set of objects (usually a subset of the entire database) used in an interactive change, add, or delete operation. Membership or lack of membership for any object in a workset does not necessarily affect the actual stored representation of an object. However, you can change or delete an object in a workset that also results in a change or deletion of the stored object. Similarly, when you add a new object (not currently stored) to a workset, the software also adds the object container.

***workspace***

Area that represents the portion of the model data needed to perform the intended task and includes the user modeling settings.

***Workspace Explorer***

Tree or list representation of objects in your workspace.

# Index

## A

- ACI • 68
- Active Template Library (ATL) • 68
- Add a load case to a load combination • 39, 42
- AISC (American Institute of Steel Construction) • 68
- AISI • 68
- analysis • 68
- analytical member • 68
- approval state • 68
- attribute • 68
- axis • 68

## B

- bay • 68
- BCSA (British Constructional Steelwork Association) • 68
- beam • 69
- bearing plate • 69
- bill of material (BOM) • 69
- boundary condition • 69
- Boundary Condition Properties Dialog Box • 66
- bulkload • 69

## C

- catalog • 69
- Catalog database • 69
- Change the analysis model member filter • 13
- CISC (Canadian Institute of Steel Construction) • 69
- Concentrated Load Properties Dialog Box • 53
- Configuration Tab • 18
- constraint • 69
- constraints • 69
- contract • 69
- coordinate • 69
- coordinate system • 70
- Create a load case • 38
- Create a load combination • 41
- Create a Mapping File • 24
- Create a new filter • 14
- Create an analysis model • 13
- cutting plane • 70

## D

- Define end releases • 66
- degree of freedom • 70
- Delete a concentrated load • 53
- Delete a distributed load • 61
- Delete a load case • 38
- Delete a load combination • 43
- Delete an analysis model • 14
- Distributed Load Properties Dialog Box • 61
- drawing tool • 70

## E

- easting • 70
- end releases • 70
- Export analysis model • 28
- Export Analytical Models • 25

## F

- fabricate • 70
- fasteners • 70
- feature • 70
- field adjustment • 70
- finite element • 70
- flavor • 71
- focus of rotation • 71
- full penetration weld • 71

## G

- General Tab (Boundary Condition Properties Dialog Box) • 67
- General Tab (Concentrated Load Properties Dialog Box) • 53
- General Tab (Distributed Load Properties Dialog Box) • 61
- General Tab (Load Case Properties Dialog Box) • 39
- General Tab (Load Combination Properties Dialog Box) • 43
- General Tab (Model Properties Dialog Box) • 16
- generic specific • 71
- GUIDs • 71

## I

- initial structural plan • 71

interference checking • 71

## K

kinematics analysis • 71

kips • 71

ksi • 71

## L

leg length analysis • 71

lintel • 71

load (structure) • 71

Load Case Properties Dialog Box • 39

Load Cases Tab (Load Combination

Properties Dialog Box) • 44

Load Combination Properties Dialog Box • 43

load group • 72

Loads • 34

logical member • 72

## M

material analysis • 72

material list • 72

material properties • 72

member name • 72

member part • 72

member system • 72

Model Properties Dialog Box • 16

Modify a boundary condition • 65

Modify concentrated load magnitude • 52

Modify distributed load magnitude • 59

Modify distributed load position • 60

Move a concentrated load • 52

move from point • 72

move to point • 72

MTO neutral file • 72

## N

New Analysis Models • 11

New Load Cases • 37

New Load Combinations • 40

New Mapping File • 21

node • 72

northing • 73

Notes Tab • 19

## O

object • 73

occurrence (of part or equipment) • 73

occurrence property • 73

origin • 73

origin point • 73

orthogonal • 73

orthographic • 73

orthotropic material • 73

## P

Place a fully distributed load • 57

Place a partially distributed load • 58

Place concentrated load by absolute key-in • 47

Place concentrated load by relative key-in • 48

Place concentrated load using point along • 50

Place fixed boundary condition • 65

Place New Concentrated Loads • 45

Place New Distributed Loads • 55

Place pinned boundary condition • 65

Preface • 6

Preview analysis/design results • 33

principle of superposition • 73

## R

reference data • 74

Relationship Tab • 17

Remove a load case from a load combination • 42

Removing a boundary condition • 66

route • 74

## S

Selecting Objects • 9

Set Boundary Conditions • 63

stress • 74

Structural Analysis • 7

Structural Analysis Common Tasks • 8

Structural Analysis Workflow • 8

structure analysis • 74

stud • 74

suspended floor • 74

symmetric node • 74

system • 74

## T

target point • 74

tolerant geometry • 74

Transfer Ownership Dialog Box • 19

trimmed surface • 74

trunk • 75

**U**

- unit/module modeler • 75
- Update an analysis model • 13
- Update analysis model using  
analysis/design results • 33
- Update from Analysis/Design • 30
- user attributes • 75

**V**

- vertex • 75

**W**

- weight and CG analysis • 75
- welding • 75
- What's New in Structural Analysis • 6
- working plane • 75
- workset • 75
- workspace • 75
- Workspace Explorer • 75